

Application of the Volume of Fluid Method with Heat Transfer to a Two- Shaft Aero-Engine Bearing Chamber

Andrea Bristot

BEng, MSc

Thesis submitted to The University of Nottingham
for the degree of Doctor of Philosophy

August 2017

Abstract

The innovations proposed for future civil aero engine concepts are likely to pose new challenges for the oil system. Higher cooling requirements and smaller core sizes mean that an effective use of oil is fundamental to secure the efficiency gains achievable with new designs. The introduction of accurate tools to predict the behaviour of air and oil flows is key to support this process.

Over the past 15 years, several Computational Fluid Dynamics (CFD) techniques have been developed to simulate elements of the air and oil flows in oil system cavities. This process has been incremental and prior to the work presented in this thesis a demonstration of the full application onto a real geometry had not been attempted. This step is necessary to evaluate the challenges still to be addressed for a complete applicability, as well as the relative relevance of the various modelling aspects. Within this framework, this thesis implements the Volume of Fluid (VOF) method for the first time onto a real bearing chamber geometry, namely the Rolls-Royce BR725 Rear Bearing Chamber.

In this work, a preliminary study is performed on a geometrically simplified bearing chamber geometry for which previous experimental data exists to identify the optimal settings and possible challenges before progressing to the real application. The importance of the liquid-gas interface is identified and different interface reconstruction techniques and turbulence damping approaches are tested and developed on this configuration. The results show the relevance of turbulence damping in influencing the flow regime, identifying aspects that could not be predicted from the stratified channel flow tests that have informed previous turbulence damping methodologies.

The implementation on the BR725 builds on these results, additionally introducing polyhedral meshing. Two operating conditions representative of the spectrum of flow regimes are simulated. A wall-fluid heat transfer model is included. The results show good agreement with the design assumptions and experimental data, in particular for flow conditions associated with the “ground idle” engine operating condition. At chamber conditions associated with high power engine operation, the gas-liquid interaction modelling limitations identified in the preliminary study emerge and are investigated. Despite the higher degree of uncertainty on these results, the comparison with the experimental data available shows that a reasonable accuracy is achieved, having selected a suitable level of turbulence damping. The data obtained from the CFD model identified a

link between the oil behaviour in the bearing chamber annulus and the frequency content of the scavenge flows.

The work presented in this thesis represents a significant step forward in bearing chamber modelling capability using the VOF method. The results obtained for the BR725 rear bearing chamber will support future research activities as well as design investigations on current bearing chamber problems.

Acknowledgements

The research presented in this thesis has received funding from the People Programme (Marie Curie Actions) of the European Union's Seventh Framework Programme (FP7/2007-2013) under REA grant agreement no. 608322 (INNOVATE Project). The work has been carried out at the University of Nottingham's Gas Turbine and Transmissions Research Centre (G2TRC). The Centre provided all the facilities, support, technical skills and significant computational resources needed. This thesis has been done in collaboration with Rolls-Royce Deutschland Ltd and Co KG (RRD), which is acknowledged for sharing the experience of its specialists and all the necessary data. The Company also supported my INNOVATE Project-funded secondment at its facility in Dahlewitz, Germany. On top of being a great experience from a personal development point of view, the secondment has been key to align the work performed with the perspective of the oil system designers, supporting the scope of this thesis.

I sincerely acknowledge the contribution of my supervisors, Prof Hervé Morvan and Dr Kathy Simmons, in providing a solid guidance in my PhD. I am grateful to Dr Michael Klingsporn, Dr Thierry Sibilli and Jens Klostermann for the useful contributions, in particular during my secondment in Dahlewitz. I would like to thank also Dr Wolfram Kurz, who discussed with me his experimental findings, and Mike Walsh, for his interest in my work. I am grateful to my internal assessor, Prof Adam Clare, for supporting my work planning in the first and second year review.

The team of researchers at the G2TRC has been of great help during my progression. I am thankful in particular to Dr Bruce Kakimpa, Dr Adeniyi Akinola, Dr Thiago Piazero De Carvalho and Dr Kiran Kumar for sharing their vast knowledge on CFD and meshing. A special mention goes to Gerry Salerno, for his support on several occasions.

In the INNOVATE Project, I would like to thank all the colleagues in my cohort for sharing these years together, both inside and outside the academia. I am also grateful to Prof Ernesto Benini and Prof Luisa Rossetto at the University of Padova, as well as Dr Bobby Sethi and Dr Eduardo Anselmi at Cranfield University, for helping me start this journey.

Finally, I wish to thank my whole family, in particular my parents, for their education and for constantly supporting me in my studies. A great thanks goes to Sarah, for being always next to me, in particular during the writing-up of this work.

List of publications

Published

Bristot, A., Morvan, H.P., and Simmons, K.A., 2016, "Evaluation of a Volume of Fluid CFD Methodology for the Oil Film Thickness Estimation in an Aero-engine Bearing Chamber", Paper GT2016-56237, Proceedings of ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, Seoul, South Korea.

Bristot, A., Morvan, H.P., Simmons, K.A., and Klingsporn, M., 2017, "Effect of Turbulence Damping in VOF Simulation of an Aero-Engine Bearing Chamber", Paper GT2017-63436, Proceedings of ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, Charlotte, NC, USA.

Ahumada Sanhueza, C., Bristot, A., Kumar, S., Schneider, N., Garvey, S., and Morvan, H., 2016, "An Integrated System's Approach Towards Aero Engine Subsystems Design", Paper SAE 2016-01-2020.

Rolls-Royce Internal Reports

Summary of Research Project INNOVATE on Bearing Chamber CFD, 24/05/2017, EDNS01000555331.

Planned

Evaluation of VOF Turbulence Damping Methods for the Simulation of Aero-Engine Bearing Chambers, *Journal of Turbomachinery*, expected submission in December 2017.

Contents

Abstract.....	i
Acknowledgements.....	iii
List of publications	iv
List of Figures.....	viii
List of Tables	x
Nomenclature.....	xi
Acronyms.....	xi
Symbols.....	xii
Subscripts.....	xiv
1 Introduction	1
1.1 Background.....	1
1.2 Overview of the Aero-Engine Oil System	4
1.3 Bearing Chambers Thermal and Fluid Design Challenges	5
1.4 Aims and Objectives	7
1.5 Thesis Overview	8
2 Literature Review.....	10
2.1 Introduction	10
2.2 Experimental Activities on Bearing Chambers.....	10
2.2.1 Introduction	10
2.2.2 Heat Transfer	11
2.2.3 Oil Film Distribution	14
2.2.4 Air Flows	15
2.2.5 Off-takes Design and Scavenge Efficiency	17
2.3 Numerical Modelling.....	18
2.3.1 Overview.....	18
2.3.2 Thin Film Model (TFM).....	19
2.3.3 Smoothed Particle Hydrodynamics (SPH)	22
2.3.4 Volume of Fluid (VOF) Method.....	24
3 Proposed VOF Modelling Approach.....	35
3.1 Introduction	35
3.2 Flow Equations	35
3.2.1 Overview.....	35
3.2.2 Continuity Equation.....	36
3.2.3 Momentum Equation	36
3.2.4 Energy Equation.....	36

3.3	The Volume of Fluid (VOF) Model.....	37
3.3.1	Overview	37
3.3.2	Implicit Formulation of the Volume Fraction Equation	38
3.3.3	Explicit Formulation of the Volume Fraction Equation	38
3.3.4	Interface Computation	39
3.4	Turbulence Modelling	43
3.4.1	The Reynolds-Averaged Navier-Stokes (RANS) Equations.....	43
3.4.2	Standard k - ϵ Model.....	46
3.4.3	RNG k - ϵ Model.....	48
3.4.4	k - ω Model.....	49
3.4.5	SST k - ω Model	51
3.4.6	Near-wall Modelling: Momentum Transfer	52
3.4.7	Near-wall Modelling: Heat Transfer.....	56
3.5	Turbulence Damping.....	59
3.5.1	Overview	59
3.5.2	Egorov's Turbulence Damping	59
3.5.3	Hashmi's Mathematically Closed Interface Treatment.....	61
3.6	Discretisation and Solution Method.....	63
3.6.1	Introduction	63
3.6.2	The Finite Volume Method.....	63
3.6.3	Solution Algorithms.....	67
4	Validation: KIT Bearing Chamber.....	69
4.1	Introduction.....	69
4.2	Set-up	70
4.2.1	Bearing Chamber Geometry.....	70
4.2.2	Meshes	71
4.2.3	Operating Conditions.....	72
4.2.4	Boundary Conditions.....	72
4.2.5	Turbulence Models	73
4.2.6	Solver Settings	74
4.2.7	Film Thickness Averaging	75
4.2.8	Set-up Summary	78
4.3	Single-Phase Results	79
4.4	Two-Phase Results.....	82
4.4.1	Evaluation of Interface Computation Methods	82
4.4.2	Baseline Case.....	84
4.4.3	Oil Films Mesh Independence Study	89
4.4.4	Effect of Turbulence Damping.....	91
4.5	Heat Transfer.....	100
4.6	Summary and Discussion of the Results.....	103
4.7	Simulation Guidelines.....	104
5	Application: Rolls-Royce BR725 Rear Bearing Chamber.....	108
5.1	Introduction.....	108
5.2	Objectives.....	108
5.3	Set-up	109
5.3.1	Bearing Chamber Geometry.....	109
5.3.2	Mesh.....	111

5.3.3	Operating Conditions	114
5.3.4	Boundary Conditions	117
5.3.5	Solver Settings.....	118
5.3.6	Simulation Strategy.....	120
5.3.7	Set-up Summary.....	121
5.4	Single-Phase Results	123
5.5	Two-Phase Results	125
5.5.1	Oil Residence Volume	125
5.5.2	Oil Outlet Flows and Scavenge Efficiency	126
5.5.3	Frequency Content of Oil Outlet Flows	129
5.5.4	Oil Film Velocities.....	133
5.5.5	Film Thickness Distribution	135
5.6	Heat Transfer	139
5.6.1	Set-up	139
5.6.2	Results at Idle.....	140
5.6.3	Results at MTO.....	143
5.7	Summary and Discussion of Results	145
5.8	Simulation Guidelines	147
6	Conclusions	150
6.1	Summary	150
6.2	Contributions to Knowledge	151
6.3	Recommendations for Future Work.....	151
7	Literature.....	154

List of Figures

Figure 1.1 – Key players in Technology by TRL score and investment.....	2
Figure 1.2 – Positioning of this thesis work.....	3
Figure 1.3 – Rolls-Royce Trent 500 oil system schematic	4
Figure 1.4 – HP/IP structure on a three-shaft engine	5
Figure 1.5 – Bearing chamber schematic.....	7
Figure 2.1 – KIT high speed bearing chamber test rig setup	12
Figure 2.2 – Heat transfer dependency on operating conditions.....	13
Figure 2.3 – Possible bearing chamber flow regimes	14
Figure 2.4 – Bearing chamber secondary air flow regimes	16
Figure 2.5 – Typical film thickness in a bearing chamber section	20
Figure 2.6 – SPH simulations of aero engine oil systems	23
Figure 2.7 – Illustration of surface reconstruction approaches	25
Figure 2.8 – Example of gas-liquid stratified flow	26
Figure 2.9 – Enhancement of velocity profiles prediction in VOF.....	28
Figure 2.10 – Effect of gas-liquid interface treatment on oil recirculation	29
Figure 2.11 – Effect of mesh refinement on oil distribution	30
Figure 2.12 – Bearing chamber simulation using adaptive meshing	31
Figure 2.13 – Schematic of DPM-VOF approach.....	32
Figure 2.14 – Results with DPM-VOF approach	33
Figure 2.15 – TFM-VOF concept.....	34
Figure 2.16 – Application of TFM-VOF on bearing chamber	34
Figure 3.1 – Implementation of coupled level-set VOF method	42
Figure 3.2 – Energy spectrum of turbulence in a grid.....	45
Figure 3.3 – Boundary layer structure.....	53
Figure 3.4 – Velocity distribution in proximity of a solid wall.....	54
Figure 3.5 – Momentum and heat exchange in turbulent boundary layer.....	56
Figure 3.6 – Effect of interface turbulence damping on the velocity profiles	61
Figure 3.7 – Control volumes for discretisation of scalar transport equation	65
Figure 3.8 – Pressure-based solution algorithms.....	68
Figure 4.1 – KIT bearing chamber experimental setup	70
Figure 4.2 – Cross-section of KIT bearing chamber mesh	71
Figure 4.3 – Location of film thickness sensors in KIT bearing chamber.....	77
Figure 4.4 – Numerical and experimental oil film thickness measurement areas	77

Figure 4.5 – Location of section plane at $\varphi=270^\circ$	79
Figure 4.6 – Air velocity vectors and plots in section plane $\varphi = 270^\circ$	81
Figure 4.7 – Example of instantaneous air velocity vectors in section plane $\varphi = 270^\circ$.	81
Figure 4.8 – Time evolution of oil film thickness, modified HRIC scheme	83
Figure 4.9 – Time evolution of oil film thickness, Compressive reconstruction.....	83
Figure 4.10 – Oil residence volume against flow time in KIT bearing chamber.....	85
Figure 4.11 – Iso-surface of oil volume fraction in baseline case	86
Figure 4.12 – Time-averaged circumferential oil film thickness distributions	87
Figure 4.13 – Time evolution of vent and scavenge pipe oil mass flow rates, baseline....	88
Figure 4.14 – Near-wall mesh refinement approach.....	90
Figure 4.15 – Mesh independence study.....	90
Figure 4.16 – Time evolution of oil residence volumes	93
Figure 4.17 – Oil film distribution for different turbulence damping approaches.....	94
Figure 4.18 – Numerical and experimental oil film thickness distributions	95
Figure 4.19 – Contours of oil film surface velocities at $\alpha_0 = 0.5$, time-averaged over 0.1 seconds	97
Figure 4.20 – Velocity vectors and oil distribution in cross-section.....	97
Figure 4.21 – Time-evolution of bearing chamber outlet temperatures	101
Figure 4.22 – Bearing chamber thermal simulation results, contour plots on heated walls; (a) Heat transfer coefficients, (b) Static temperatures, (c) Oil volume fractions.....	102
Figure 4.23 – KIT bearing chamber context diagram	105
Figure 5.1 – CAD model of Rolls-Royce BR725 Rear Bearing Chamber	109
Figure 5.2 – CFD model of BR725 Rear Bearing Chamber	111
Figure 5.3 – Bearing chamber polyhedral mesh	113
Figure 5.4 – Cross-section of volume mesh in xy plane.....	114
Figure 5.5 – Contours of cell convective Courant numbers (CFL)	120
Figure 5.6 – BR725 RBC simulation procedure	121
Figure 5.7 – Time-evolution of shear stress integrals	124
Figure 5.8 – Time-evolution of oil residence volume in bearing chamber annulus	125
Figure 5.9 – Contours of static pressure in bearing chamber cross-section.....	126
Figure 5.10 – Location of monitor planes for oil outlet flows	127
Figure 5.11 – Time-evolution of oil flow rates in scavenge pipe	128
Figure 5.12 – Time-evolution of oil flow rates in vent pipe	128
Figure 5.13 – Frequency content (FFT) of oil outlet flows at ground idle.....	131
Figure 5.14 – Frequency content (FFT) of oil outlet flows at MTO.....	132
Figure 5.15 – Contours of oil film surface tangential velocities at $\alpha_0 = 0.5$	134

Figure 5.16 – Visualisation of recirculation zone generated by jetblock	135
Figure 5.17 – Location of planes for oil film thickness computation	136
Figure 5.18 – Idle oil circumferential film thickness distributions	137
Figure 5.19 – MTO oil circumferential film thickness distributions	137
Figure 5.20 – Time-evolution of average annulus temperature at idle.....	141
Figure 5.21 – Contours of wall heat transfer coefficients (a, b), and contours of wall volume fractions (c, d) at ground idle.....	142
Figure 5.22 – Time-evolution of average annulus temperature at MTO.....	143
Figure 5.23 – Contours of wall heat transfer coefficients (a, b), and contours of wall volume fractions (c, d) at MTO.....	144

List of Tables

Table 4.1 – KIT bearing chamber dimensions	71
Table 4.2 – Operating conditions and fluid properties	73
Table 4.3 – General two-phase simulation settings	78
Table 4.4 – Turbulence damping cases tested	92
Table 4.5 – Computed liquid quantities.....	93
Table 4.6 – Effect of turbulence damping on scavenge efficiencies	94
Table 4.7 – Turbulent quantities computed in bearing chamber annulus	98
Table 5.1 – Rear Bearing Chamber relevant dimensions	110
Table 5.2 – Mesh characteristics	112
Table 5.3 – Fluid and thermal operating conditions.....	115
Table 5.4 – Material properties	116
Table 5.5 – Time step limits for $CFL_{\max}=1$ near seals and final chosen time step.....	119
Table 5.6 – BR725 RBC two-phase simulation settings.....	122
Table 5.7 – FFT sampling characteristics.....	130
Table 5.8 – Thermal boundary conditions and material properties	140
Table 5.9 – RBSS simulation computational costs.....	147

Nomenclature

Acronyms

CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CLSVOF	Coupled Level-Set VOF
CICSAM	Compressive Interface Capturing Scheme for Arbitrary Meshes
DES	Detached-Eddy Simulation
DPM	Discrete Phase Model
FEM	Finite Element Method
FFT	Fast Fourier Transform
FVM	Finite Volume Method
G2TRC	Gas Turbine and Transmissions Research Centre
HP	High Pressure
HRIC	High Resolution Interface Capturing
HT	Heat Transfer
HTC	Heat Transfer Coefficient
KIT	Karlsruhe Institute of Technology
LES	Large Eddy Simulation
LP	Low Pressure
MCIT	Mathematically Closed Interface Treatment
MTO	Maximum Take-Off
PLIC	Piecewise Linear Interface Construction
PRESTO	Pressure Staggering Option
RBC	Rear Bearing Chamber
RNG	Re-Normalisation Group
SGS	Sub-Grid-Scale
SIT	Simplified Interface Treatment
SMD	Sauter Mean Diameter
SPH	Smoothed Particle Hydrodynamics
SR	Scavenge Ratio
SST	Shear Stress Transport
TFM	Thin Film Model
TKE	Turbulent Kinetic Energy
TRL	Technology Readiness Level
URANS	Unsteady Reynolds-Averaged Navier-Stokes
UTC	University Technology Centre
VOF	Volume of Fluid

Symbols

Symbol	Unit	Description
$\varnothing$	m	Diameter
α	-	Volume fraction
β	-	Closure coefficient of destruction term in k- ω model
	-	Bounding factor in time integration
Γ	-	Diffusion coefficient
γ	degrees ($^{\circ}$)	Inlet swirl angle
Δn	m	Near-wall cell height
ϵ	m ² /s ³	Turbulence kinetic energy dissipation rate
δ	m	Boundary layer depth
δ_{ij}	-	Kronecker delta
η	-	Efficiency
κ	m	Wave number
	-	Karman constant
	m ⁻¹	Interface curvature
Λ	-	Activation function of MCIT
λ	m	Eddy wave length
	-	Gravity parameter
μ	Pa·s	Dynamic viscosity
ν	m ² /s	Kinematic viscosity
ρ	kg/m ³	Density
σ	-	Turbulent Prandtl number
	N/m	Surface tension
τ	Pa	Shear stress
Φ	-	Generic flow quantity
ϕ	m	Level-set function
φ	degrees ($^{\circ}$)	Bearing chamber angular position
ω	s ⁻¹	Turbulent kinetic energy specific dissipation rate
A	m ²	Area
	m ⁻¹	Interface area density
a	m	Interface thickness in level-set function
	-	Linearised coefficient for transport equations
B	-	Turbulence damping factor
b	m	Chamber axial length
c_p	J/(kg·K)	Specific heat at constant pressure
D_h	m	Hydraulic diameter
E	J	Energy
F		Annulus liquid filling fraction
F_s	N	Surface tension force

G	$\text{kg}/(\text{m}\cdot\text{s}^3)$	TKE generation
g	m/s^2	Gravitational acceleration
H	-	Heaviside function
h	m	Thickness
J	N	Total momentum transfer rate from the shaft to the air flow
k	$\text{W}/(\text{m}\cdot\text{K})$ m^2/s^2	Thermal conductivity Turbulence kinetic energy
l	m	Length
M	-	Mach number
$\dot{m}$	g/s	Mass flow rate
N	min^{-1} (rpm)	Rotational speed
Nu	-	Nusselt number
P	m Pa	Perimeter length Absolute pressure
Pr	-	Prandtl number
p	Pa	Relative pressure
q	W	Heat flux
R	-	Velocity ratio
Re	-	Reynolds number
r	m	Radius
S	-	Source term
S_{ji}	s^{-1}	Mean strain rate tensor
T	$^{\circ}\text{C}, \text{K}$	Temperature
T_c^*	-	Non-dimensional convective-conductive temperature
t	s	Time
U	m/s	Flow velocity
u'	m/s	Fluctuating component of flow velocity
u^+	-	Dimensionless velocity
u	m/s m/s	Wall velocity x-component of velocity
V	m^3	Volume
$\dot{V}$	m^3/s	Volume flow rate
v	m/s	y-component of velocity
w	m/s	w-component of velocity
$\mathbf{X}$	m	Initial position vector in Lagrangian approach
$\mathbf{x}$	m	Position vector in Eulerian and Lagrangian approach
Y	$\text{kg}/(\text{m}\cdot\text{s}^3)$	TKE dissipation
y^+	-	Dimensionless wall distance
$\hat{y}^+$	-	Scalable wall function y^+

Subscripts

0	Reference
∞	Free stream
α	Phase
ε	Turbulence kinetic energy dissipation rate
τ	Shear
ω	Turbulent kinetic energy specific dissipation rate
A	Acceptor
a	Air
ann	Annulus
ax	Axial
b	Buoyancy
C	Cell
ch	Chamber
D	Donor
D_h	Hydraulic diameter
eff	Effective
exp	Experiment
F	Face Oil flow
f	Film
g	Gas
h	Heat Hydraulic
I	Turbulent intensity
i	i-th direction of Cartesian axis i-th phase
in	Inlet
k	Turbulence kinetic energy
L	Sealing air flow
l	Liquid
lam	Laminar
m	Momentum
max	Maximum
min	Minimum
mix	Mixture
mol	Molecular
nb	Neighbouring cell
o	Oil
out	Outlet

<i>p</i>	At constant pressure
<i>sc</i>	Scavenge
<i>sh</i>	Shaft
<i>rw</i>	Outer (rim) wall
$\perp$	Tangential
<i>T</i>	Thermal
<i>t</i>	Turbulent
<i>U</i>	Upwind
<i>u</i>	Shaft speed
<i>vt</i>	Vent
<i>w</i>	Wall

1 Introduction

1.1 Background

In current production aero engines, the operating conditions encountered by the oil can be particularly harsh, with metal temperatures bringing it close to its coking point (typically, the metal temperature limit is 225°C for a commonly used commercial oil [1, 2]). Oil flows are critical to ensure lubrication and cooling of the transmission components, thus an effective oil system design is fundamental for a safe and efficient operation of the engine. Typically, designs rely on flows much larger than the amount needed for pure lubrication, in order to prevent oil overheating. This approach, however, takes its toll on the engine performance:

- Added weight;
- Power consumption of the oil pumps;
- Power consumption of the air-oil separators;
- Volume occupied by the system;
- Amount of oil vented overboard.

The introduction of novel architectures such as a power gearbox (PGB) connecting the low pressure turbine and the fan, or a wider electrification of the engine subsystems, will generate a significant amount of heat that will be picked up by the oil. This is likely to be of the order of tenths of times higher than in current production engines (~ 0.3 MW for a large civil engine [3]). Higher thermal cycle pressures and temperatures will lead to smaller engine core sizes, reducing the surface for heat exchange and at the same time increasing the wall temperatures encountered by the oil. Therefore, the oil system will face significant challenges to cope with the new conditions while keeping an acceptable size. This will imply a necessary step change in the oil system layout and sizing method, calling for new design methodologies to be introduced.

Facing the challenges above, as well as the need to keep a fast development pace, oil system designers aim to move towards a predictive approach based on numerical methods. The objective is to identify the optimal layout for components such as bearing chambers or gearboxes prior to the experimental validation, saving time and minimising the risk of redesigning parts by achieving a “right first time” design. The improvement of predictive

capabilities will enhance the efficient use of oil flows, reducing the oil system weight and the factors of uncertainty linked with the current methods.

Over the last 15 years, significant progress has been made towards a predictive approach for the oil system. Large steps in the development of Computational Fluid Dynamics (CFD) methods to model components such as gearboxes, bearing chambers, or de-aerators, have led to address several challenges to reproduce the flow physics correctly. The process followed up to this stage, however, has been always aimed at solving specific modelling issues, as illustrated in Figure 1.2.

An overall vision of the performance on a “real life” case, i.e. of industrial application interest, has been missing so far, mainly due to the need to assess specific modelling aspects first, as well as because of the lack of computational power available. In the last five years, however, improvements in methods such as the Volume of Fluid (and a greater availability of computational power) have suggested that this kind of application is about to reach maturity. These conditions provide the opportunity for the first time to scale up the level of the challenge, doing a comprehensive assessment of the current CFD capability to model a “real life” scenario. Following this route, this thesis aims to demonstrate the application of the VOF method to a real aero engine bearing chamber, including heat transfer, to benchmark its level of maturity against a real industrial problem.

The findings of this work are set to direct the necessary future modelling and experimental activities on bearing chambers, identifying the low Technology Readiness Level (TRL) developments required to bridge the gap between fundamental research and its applicability for industrial purposes, as illustrated in Figure 1.1.

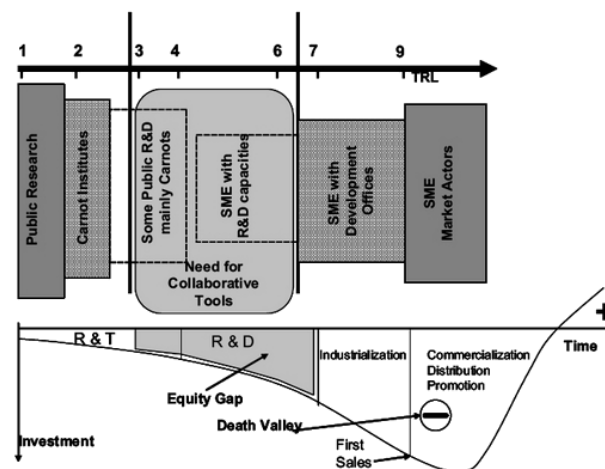


Figure 1.1 – Key players in Technology by TRL score and investment [4]

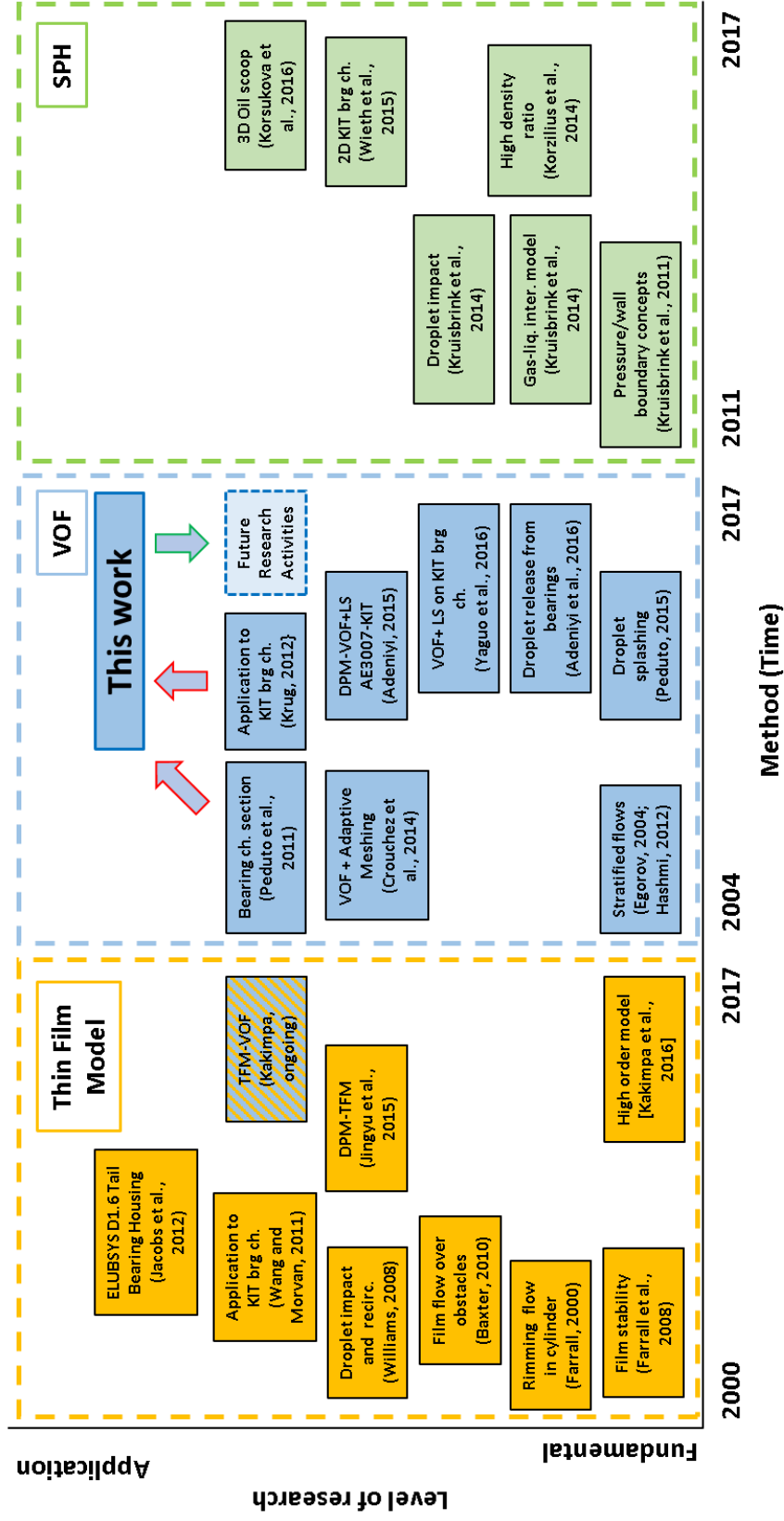


Figure 1.2 – Positioning of this thesis work within the framework of bearing chamber numerical modelling

The outcome will provide also a valuable insight for the oil system designers, as the numerical simulations will give for the first time a full picture of the fluid and thermal phenomena taking place in the bearing chamber analysed.

1.2 Overview of the Aero-Engine Oil System

In a jet engine, a constant flow of oil normally supplies all the components of the mechanical transmission requiring lubrication and cooling. These include shaft bearings, gears, vibration dampers, hydraulic seals, as well as shaft splines in some engine models. The schematic of Figure 1.3 reports a typical aero-engine lubrication system. Following the circuit starting from the tank, oil is extracted and pressurised by a volumetric pump. Downstream of the pump, it is cooled in a fuel-oil heat exchanger, after which it is delivered to the target components. Normally this stage relies on oil jets, although other forms of injection such as feed holes in the bearing races are common features. The used oil is then collected in bearing chambers or other pressurised cavities and extracted by scavenge pumps. These deliver the oil to a deaerator, namely a device separating the entrained air from the liquid, after which it re-enters the tank.

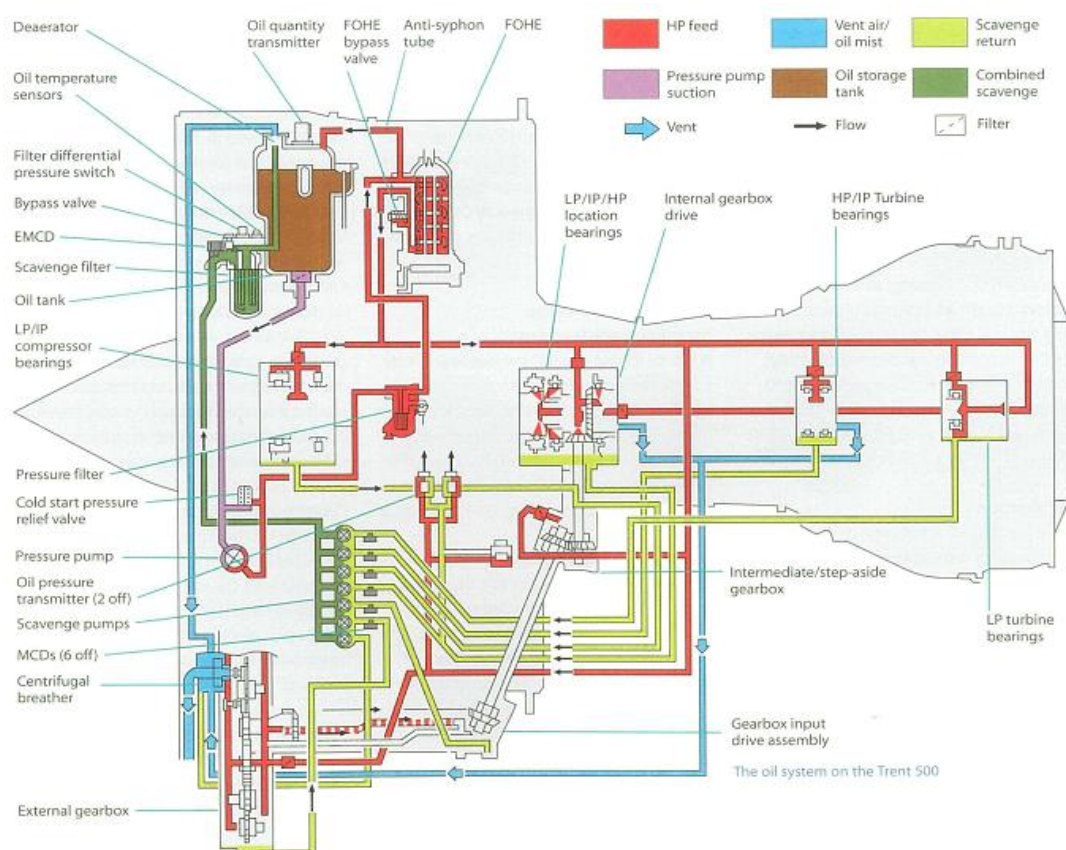


Figure 1.3 – Rolls-Royce Trent 500 oil system schematic [5]

In bearing chambers, part of the used oil is collected by the air system due to the mixing of oil droplets with the sealing air flows. In order to recover this fraction of oil, a centrifugal device called breather separates the droplets from the air, and feeds the oil back to the tank. The remaining air flow with low oil content is then vented overboard.

Oil serves also as a “health monitoring” tool for the engine, as it washes off abraded metal particles and other dirt particles that can originate in the engine transmission system. For this reason, the oil system contains magnetic chip detectors and multiple filters, so that the early signs of excessive wear or abnormal temperatures can be detected rapidly to prevent damage.

1.3 Bearing Chambers Thermal and Fluid Design Challenges

In the oil system fluid and thermal management, bearing chambers play an influential role as they are the component in which most of the heat exchange between the engine core structure and the oil takes place. Bearing chambers are characterised by an annular section surrounding the shafts, sealed with pressurised air on its extremities to prevent oil leakages between the shafts and the static walls. A scavenge pipe is located at the bottom for oil extraction, while one or multiple vent pipes are located at the top to equalise pressure.

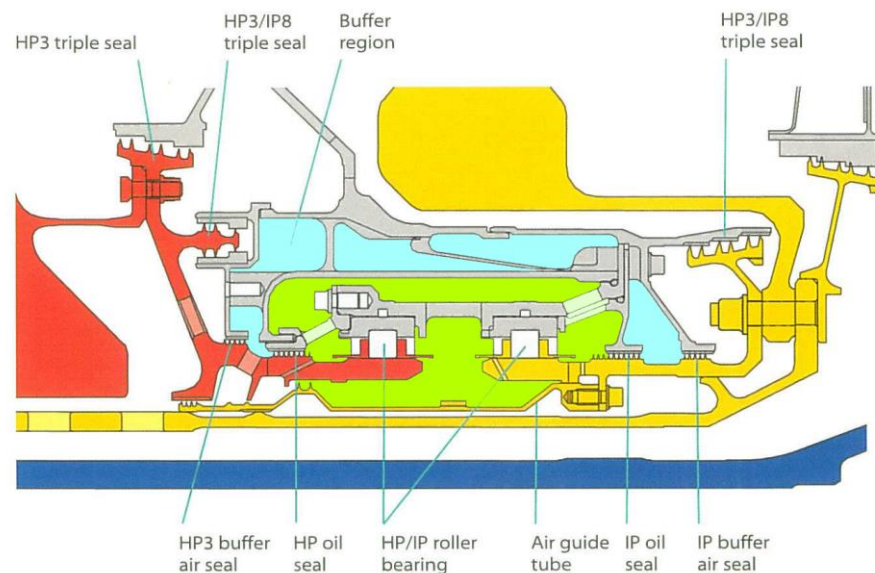


Figure 1.4 – HP/IP structure on a three-shaft engine, adapted from [5]. Green: oil-wetted cavity; cyan: buffer air region; red: HP shaft; yellow: IP shaft; blue: LP shaft

As the annulus needs to be sufficiently large to accommodate the oil injectors, the bearing support structures, as well as the oil seals and other fittings, bearing chambers offer a relatively large surface for oil to spread and exchange heat with the surrounding metal parts. Figure 1.4 shows a typical HP/IP bearing chamber arrangement for a three-shaft engine, giving an outline of the proportions between the size of the bearings and the surrounding space needed. The schematic shows that the bearing chamber is surrounded by an outer chamber pressurised by a flow of buffer air. The latter is designed to provide an inward flow in the oil-wetted chamber, as well as to insulate it thermally from the hottest engine parts.

Thermal insulation is a fundamental aspect of bearing chamber design, in particular for those in proximity of the HP turbine. The high temperatures imply that the spacing between components, position of structures, and cooling air and oil flows have an important role in the bearing chamber thermal control. While the effects of spacing and thermal conduction between structures are well understood, air and oil flow behaviours are difficult to predict. This aspect originates from the complexity of their interaction, and is highly dependent on the engine operating conditions.

Ideally, a uniform oil coverage of the bearing chamber walls is desired, as it gives homogeneous heat transfer properties and avoids the formation of hot spots where oil is not present. A uniform distribution, however, is difficult to achieve for all operating conditions. Sealing air flows and shaft windage tend to accumulate the oil in some areas of the bearing chamber, leaving other regions either covered by a thin film or dry. The gravity force is influential as well, as it leads to the build-up of a thicker layer at the bottom of the chamber, as illustrated in Figure 1.5. This behaviour is found in particular at low shaft speeds, and can lead to oil recirculation and overheating. Its numerical modelling represented for several years a considerable challenge [6-8].

The investigation of the air and oil flows is further complicated by the location of bearing chambers in the engine core, which makes it impossible to measure directly parameters like oil film thickness or air speed. Typically, flow quantities are measured at the bearing chamber outlets and are limited to air pressure and oil flow rates. Additional understanding comes from test rigs, but since these rely typically on simplified geometries, a detailed insight into the actual bearing chamber configuration is generally missing. In this perspective, numerical modelling is a powerful, low cost resource that can reproduce virtually any geometric configuration. However, as it will be seen in the following chapters,

the development of new numerical techniques in the past years relied mainly on test rig geometries for validation. Therefore, the demonstration of the current capabilities for a real-life case is still missing.

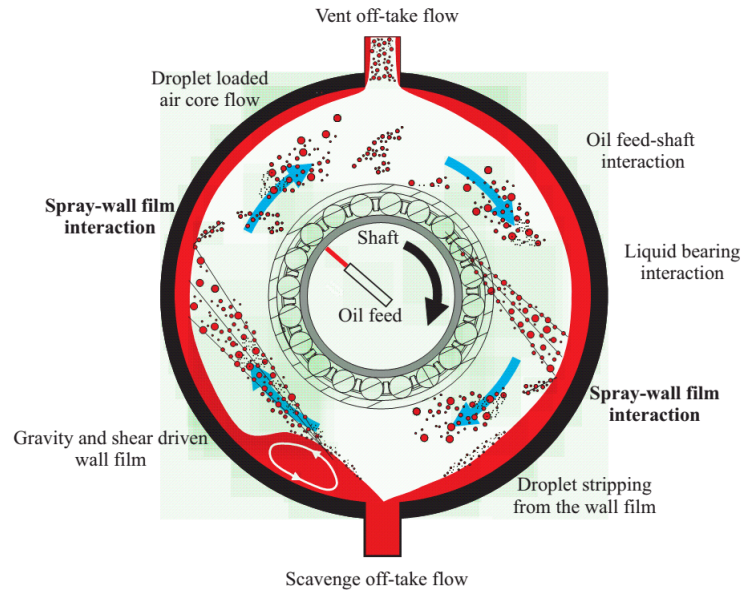


Figure 1.5 – Bearing chamber schematic showing the fluid phenomena taking place during operation [9]

To develop a tool that is useful to support bearing chamber design and optimisation, addressing the challenges described above, it is of vital importance to understand what the limitations are in the use of current modelling techniques, from both an accuracy and applicability perspective. This includes moving beyond the representation of simplified test rigs in order to assess the modelling performance on complex geometries and operating conditions. This approach will enable to concentrate future research efforts on the aspects that are critical to the application in a design context, such as computational requirements or relevant modelling inaccuracies. The scope of this thesis work is to address this level of the development process.

1.4 Aims and Objectives

Having identified the growing potential of CFD modelling to support the fluid and thermal aspects of the bearing chamber design process, it has been observed that the challenges of its implementation onto a real design case still lack a comprehensive assessment. This is due to the validation approach followed up to this stage in this research field, bound to the modelling of simple test rig cases for code validation. To address the challenge of how well current models perform on a real case, the work presented in this

thesis implements for the first time the Volume of Fluid (VOF) CFD method for the simulation of the air, oil and thermal flows on the geometry of a production aero-engine bearing chamber, under realistic operating conditions. The aims are:

1. To identify the gaps and bottlenecks of the current CFD modelling techniques;
2. To guide future research efforts onto the *relevant* aspects critical to the implementation in an industrial design environment;
3. To improve the understanding of the flow behaviour in a complex bearing chamber environment;
4. To define simulation guidelines based on the lessons learnt.

The aims listed above will be achieved through the following steps (objectives):

1. Assess the capabilities/drawbacks of existing bearing chamber CFD modelling techniques;
2. Select the most suitable model for the proposed case;
3. Evaluate the model on a simplified experimental bearing chamber;
4. Based on the lessons learnt at point 3., implement the model on the real bearing chamber geometry;
5. Evaluate the results against the available engine test and numerical data;
6. Propose follow-up modelling activities to remediate the main gaps or obstacles identified in the current method;
7. Establish guidelines for future bearing chamber simulations, based on the technique investigated.

1.5 Thesis Overview

This thesis work is divided into five main chapters. This section gives an overview of their content, summarising the methodology followed.

Chapter 2 reviews the relevant literature on bearing chamber fluid and thermal investigations. Given the complexity of the problem, past research activities relied on both experiments and numerical modelling. Among the numerical techniques available, the Volume of Fluid (VOF) method, the Thin Film model and the Smoothed Particle Hydrodynamics (SPH) method are the most common approaches found in the literature. The chapter demonstrates the current leading position of the VOF method over the other available techniques when implemented in an industrial application, and motivates its choice for the simulation work presented in this thesis. The review of the experimental

work provides further insight on the bearing chamber flow phenomena, as well as a reference case to validate the selected modelling approach.

Chapter 3 describes the relevant equations and modelling techniques adopted in this work, including turbulence modelling, a comprehensive description of the VOF method, and an overview of the solution techniques adopted. Particular attention is dedicated to turbulence damping in this chapter, as it is a critical aspect in VOF modelling of shear-driven flows such as those found in bearing chambers.

In Chapter 4, the proposed VOF approach is implemented on the reference experimental case [10]. This chapter assesses the capability of the model to reproduce the experimental film thickness and scavenge efficiency results, and identifies the factors that affect the accuracy and computational cost through the evaluation of different model settings, including the interface reconstruction scheme and the time discretisation approach. Heat transfer equations are included to assess the modelling accuracy and the additional computational costs involved. The lessons learnt have led to define the modelling choices to be implemented on the real engine case, namely the Rolls-Royce BR725 Rear Bearing Chamber (RBC).

Chapter 5 presents the modelling work on the BR725 RBC geometry. Film thickness, scavenge efficiency, outlet flows frequency content and wall heat transfer are evaluated for the two engine-representative operating conditions tested. The results are compared with the data available and the design assumptions, identifying the areas in which the model performs in good agreement with the available data, as well as those where the performance level is still not acceptable. A clear distinction is defined between the conditions and output parameters for which VOF can be trusted for design and analysis purposes, and those in which additional development work is required. With this approach, the level of maturity of the method is identified, and guidance for future fundamental research towards the full readiness of the model is provided. Recommendations are given to facilitate the set-up of an analogous model on similar engine components. These include indications on modelling choices affecting the computational times and the accuracy of the results, such as mesh sizing, time stepping, and post-processing of large volumes of data.

The conclusions of the work are presented in Chapter 6. These include a summary of the unique contributions to knowledge brought by this thesis, as well as guidelines for future work.

2 Literature Review

2.1 Introduction

This chapter reviews the research work available in the field of air-oil flows and heat transfer in aero-engine bearing chambers. Since both experimental and numerical lines of work have been actively pursued, often in complementary fashion, this chapter reflects both. It is divided into a section focusing on relevant experimental work and a section on numerical modelling. The latter specifically covers two-phase, shear-driven gas-liquid flows and their simulation in the context of bearing chamber applications. A survey of the numerical approaches commonly used in bearing chamber modelling, with examples of their application, is also included.

2.2 Experimental Activities on Bearing Chambers

2.2.1 Introduction

Over the past 20 years, several experimental campaigns have been conducted to characterise heat transfer mechanisms and oil behaviour in bearing chambers [10-32]. The experiments aimed mainly at better understanding aspects that could not be analysed through conventional engine testing, such as wall heat transfer characteristics [11-13, 16, 17] or the oil split between vent and scavenge pipes [10]. The relevant quantities have been evaluated for different geometries and operating conditions [21, 22], in order to define empirical correlations to inform bearing chamber and air/oil system sizing. One major limitation of this approach is that the correlations are restricted to the range of operating conditions and geometries tested, which do not cover the whole spectrum of bearing chamber configurations and operational envelopes normally found in an engine.

Major experimental contributors to bearing chamber research have been the University of Nottingham's Gas Turbines and Transmissions Research Centre (G2TRC)¹ [24-30] and the Karlsruhe Institute of Technology (KIT) [10-18, 20-23, 33, 34]. Additional work has been done at the University of Thessaloniki, Greece [35], and at the Northwestern Polytechnical University of Xi'an, China [31].

¹ Home to the Rolls-Royce Transmissions University Technology Centre (UTC)

The topics of the experimental campaigns in these references can be categorised in the following research areas:

- (a) Definition of wall heat transfer properties;
- (b) Characterisation of wall film speeds and thicknesses;
- (c) Air velocity measurements in the chamber annulus;
- (d) Evaluation of sump configurations and geometries (e.g. slotted, covered ramp);
- (e) Assessment of scavenge efficiency (i.e. oil split between scavenge and vent).

Based on the list above, the core areas of the experimental research at the University of Nottingham are (b), (d), and (e), while the publications from KIT cover mainly areas (a, b, c, e). The work conducted at the University of Thessaloniki provides data on air velocities in the bearing chamber (c), while the experimental activities from the Northwestern Polytechnical University of Xi'an include wall film thickness measurements, thus area (b).

In the following sections, the relevant experimental findings in each of the areas above are reported in detail.

2.2.2 Heat Transfer

The first modular test rig for bearing chamber research was introduced at the KIT by Wittig et al. [11] in 1994. The rig was designed to investigate two-phase flows in real engine conditions, reconstructing a bearing chamber geometry including preheated, pressurised sealing air and oil flows (respectively of up to 50 g/s and 12 g/s), variable shaft speeds up to 12,000 rpm, a roller bearing with squeeze-film dampers, and a transparent cover to visualise the oil behaviour inside the chamber. Figure 2.1 shows a schematic of the rig. The housing was designed mainly for heat transfer analysis, thus instrumented with wall, gas, bearing flange and chamber coolant temperature sensors. Ultrasonic film thickness sensors were used as well, having the advantage of being non-intrusive measurement devices [11].

The work of Wittig et al. [11] gave insight on the flow and heat transfer phenomena taking place in bearing chambers, demonstrating the dependency of heat transfer characteristics on the shaft rotational speed and air/oil mass flow ratios. Nonetheless, the extent of the results was still limited to one circumferential location for the heat transfer coefficients, and the observation of strong three-dimensional effects in the flow prompted for further investigations.

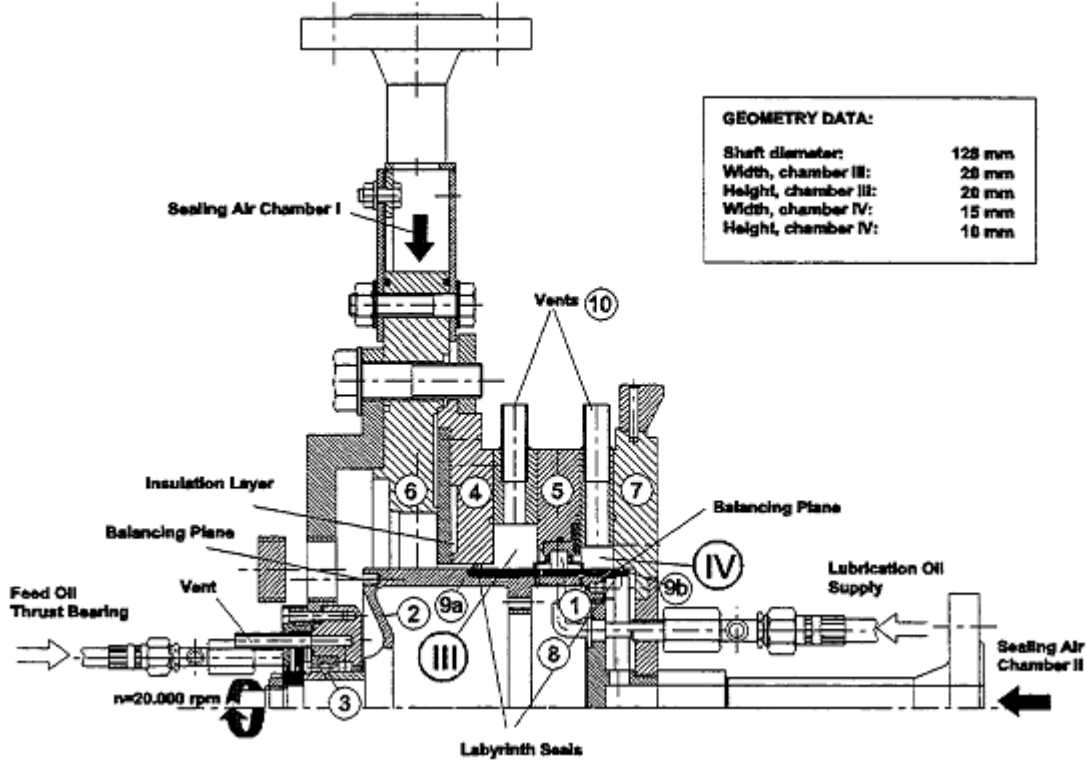


Figure 2.1 – KIT high speed bearing chamber test rig setup : (1) squeeze-film-damped roller bearing, (2) rotor, (3) ball bearing, (4) housing, (5) roller bearing support, (6) flange, (7) chamber cover, (8) under-race lubrication, (9) labyrinth seals, (10) vent [12]

The work was continued by Glahn and Wittig [15-17], who characterized the oil film flow velocities in an attempt to match an analytical model with the predictions, although deviations of the order of 30% were found [16]. The first correlations for heat transfer and film heights were derived by Chew [36] at Rolls-Royce, and Glahn and Wittig [17].

Experimental work on the effect of different bearing chamber geometries and operating conditions on wall heat transfer was then started by Glahn et al. [13] and continued by Busam et al. [12]. These experiments led to an empirical correlation for the heat transfer inside the chamber, expressed as a function of the chamber geometry, flow rates and shaft speed,

$$Nu_{D_h} = 0.35 \cdot D_h^{1.46} \cdot Re_a'^{0.48} \cdot Re_o'^{0.32} \cdot Re_u^{0.35} \quad (2.1)$$

where

$$D_h = \frac{4A}{P} \quad (2.2)$$

is the chamber hydraulic diameter (with A the area of the cross-section, P its wetted perimeter length). Re'_a and Re'_o are respectively the Reynolds numbers of the sealing air flow and the oil flow, and are computed as follows

$$Re'_a = \frac{4 \dot{m}_{ch}}{\mu_a \bar{T}_{ch} D_h} \quad (2.3)$$

$$Re'_o = \frac{4 \dot{V}_{ch}}{\nu_o \bar{T}_{ch} D_h} \quad (2.4)$$

Re_u is a circumferential Reynolds number based on the shaft speed:

$$Re_u = \frac{2 \pi N_{sh} r_{sh} D_h}{\nu_o \bar{T}_{ch}} \quad (2.5)$$

The heat transfer rate is quantified with the Nusselt number Nu , namely the ratio between convective and conductive heat transfer. Correlation (2.1) is valid for most of the operative conditions of the bearing chamber, being applicable for sealing mass flows comprised between 5 and 20 g/s, oil flows between 24 and 176 l/h, rotational speeds from 4,000 to 16,000 rpm and chamber hydraulic diameters between 0.012 and 0.034 m [12]. With (2.1), heat transfer values have been found to fall within $\pm 20\%$ of the experimental data, as shown in Figure 2.2. Moreover, looking at the exponential values of the correlation, the oil flow rates (quantified with Re'_o) have a weaker impact on heat transfer than the sealing air flow rates (Re'_a). The wall heat transfer properties also increase with the shaft rotational speed (Re_u), and hydraulic diameter (D_h).

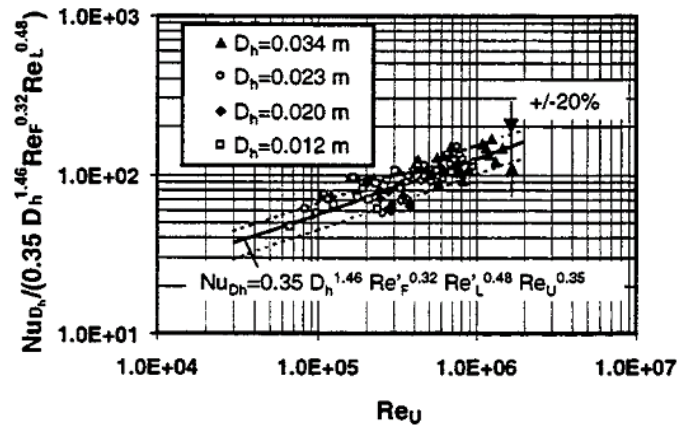


Figure 2.2 – Heat transfer dependency on operating conditions : correlation and experimental data [12]

The experiments of Busam et al. [12] have also quantified the local wall heat transfer coefficients (HTC) in the range $2000 \text{ W}/(\text{m}^2\text{K}) \leq HTC \leq 6000 \text{ W}/(\text{m}^2\text{K})$, with the

upper values achieved at the highest shaft speeds. The increase of the wall heat fluxes with the shaft rotational speed is in line with the numerical findings of Kay [37] at Nottingham, highlighting that both the higher air shear and kinetic energy of the impinging oil droplets lead to faster moving films on the wall, thus increasing the convective heat transfer capabilities of the oil film. An additional analytical approach to compute the wall heat transfer coefficients is found in the work of Glahn and Wittig [16].

2.2.3 Oil Film Distribution

While the initial bearing chamber experimental activities focused mainly on the definition of wall-fluid heat transfer correlations (see Section 2.2.2), successive work aimed at understanding how the oil film distribution on the walls inside the chamber depends on the operating conditions of the engine. This aspect plays a significant role in the wall cooling effectiveness of the oil, as regions characterised by dry-out or excessive accumulation of oil can form at certain operating conditions. The first experiments to characterise the dependency of oil distribution on the engine operating conditions were carried out by Gorse et al. [19]. As expected, the authors observed that the rotational speed affects considerably the film thickness on the bearing chamber walls. Two distinct oil flow conditions, dominated by gravity forces at low shaft speed and by air shear at high shaft speed, separated by a transition phase, were identified. Figure 2.3 illustrates qualitatively the variation of flow regime for increasing shaft speeds.

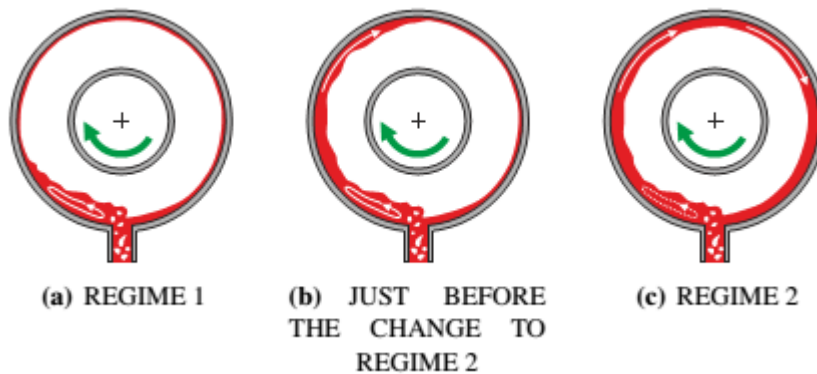


Figure 2.3 – Possible bearing chamber flow regimes : (a) Oil pooling, (b) Transition to smooth distribution, (c) Smooth distribution [20]

The point of transition between gravity-dominated and shear-dominated oil flows does not depend only on the shaft speed. It is influenced by several factors including the chamber geometry and the inlet oil and air mass flows. Kurz and Bauer [20] investigated this aspect and proposed the total momentum transfer rate J from the shaft to the air flow

as a criterion to determine the flow regime. From the experimental measurements at a constant oil temperature of 100°C [38], the authors identified a value of $J \approx 120 (Ns)s^{-1}$ in correspondence of the transition between the two regimes. A correlation for estimating J according to the shaft rotational speed, chamber axial length, pressure, and oil viscosity was proposed. This approach, however, was limited by the non-linear dependence of the flow regime change on the chamber diameter and oil flow rate. The outcome confirms that the characterisation of the flow regime in the bearing chamber with a simple correlation is particularly challenging to achieve, and suggests that CFD analysis is a more viable route to make predictions at this level. Kurz and Bauer [20] also observed that the momentum input due to droplet impact does not have a relevant effect on the oil film thickness distribution trend, as different oil inlet flow rates did not affect the latter significantly. This result confirms that air shear is the major driver of the oil film, and indicates that appropriate modelling of the air-oil interaction is a critical aspect for the accuracy of any bearing chamber CFD analysis. For this reason, air-oil interaction modelling is one of the main topics on which this thesis work will focus.

Additional experimental data on the effect of geometry variations on the oil film thickness distribution is found in the work of Kurz et al. [10, 21, 22], while Hashmi et al. [32] investigated the air pressure drop in a rectangular channel for different operating conditions to characterise the gas-liquid interaction in bearing chambers. Chandra et al. [28] at Nottingham, and Jingyu and Zhenxia [31] also performed oil film thickness measurements in bearing chambers, although the latter two works are limited to circumferential locations that cover only part of annulus. Therefore, the most comprehensive validation dataset for this thesis work has been identified in the work of Kurz et al. [10].

2.2.4 Air Flows

As stated in the discussion of Equation (2.1), sealing air flows have a significant impact on the heat transfer properties of the bearing chamber. The axial velocity component of the sealing air entering the chamber tends to generate secondary vortices, i.e. in the radial planes of the annulus. These vortices can affect the oil distribution, typically leading to accumulations on the axial extremities of the annulus. From the measurement of the air velocity distribution in the secondary vortices, Gorse et al. [18] identified two regimes which depend on the shaft speed and sealing air flow rate. According to [18], the vortices are driven either by the axial momentum of the sealing air injected in the chamber (sealing

air driven mode) or by the shaft rotational speed (rotational speed driven mode), with a transition regime dividing the two cases. In the first case, normally a large, single vortex is found, while in the second case two counter-rotating vortices tend to form. Figure 2.4 illustrates the two air flow regimes.

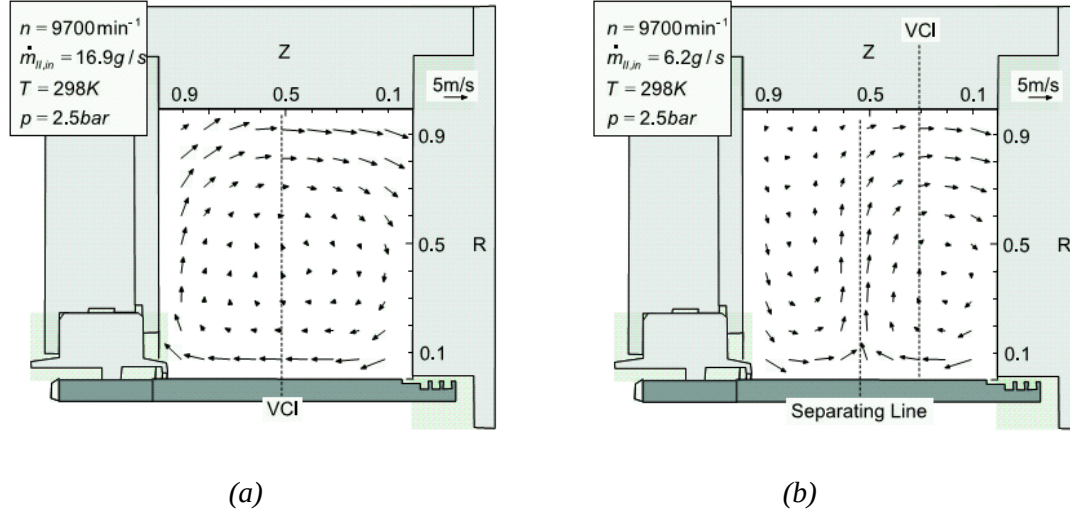


Figure 2.4 – Bearing chamber secondary air flow regimes : (a) Sealing air driven mode, (b) Rotational speed driven mode [18]

The findings of Gorse et al. [18] led to the definition of a velocity ratio to predict the expected flow regime. The ratio is defined as follows

$$R_v = \frac{u_{sh}}{U_{ax,in}} \quad (2.6)$$

where u_{sh} is the shaft surface velocity and $U_{ax,in}$ the inlet axial component of the sealing air velocity. Values above $R_v = 4.2$ were found to give rotational speed driven secondary vortices, while lower values yield sealing air dominated vortices [18]. As the latter correlation was derived from the dataset of a single geometry [18] however, its applicability to different bearing chamber configurations may be limited. Nonetheless, the results outline the implications that a different choice of seals with lower leakage rate (e.g. carbon seals, brush seals) may have on the secondary air flow, and suggest that the oil film distribution may change consequently.

Additional measurements of the air velocity distributions in the chamber annulus cover the purely axial components and are found in the work of Aidarinis et al. [35]. This experiment identified recirculation zones in the axial direction, analogously to what found by Gorse et al. [18]. The results were used to validate a CFD modelling approach based on the RSM turbulence model, finding good agreement between the experimental and

numerical data. This suggests that CFD is suitable to obtain accurate predictions of the air flows in a dry bearing chamber. However, the introduction of oil adds significant modelling challenges and uncertainties, as will be discussed in the following chapters.

The results of Aidarinis et al. [35] confirmed the recirculating behaviour previously found by Gorse et al. [18], with the main differences being a taller bearing chamber cross-section and relatively low shaft speeds (between 4,000 and 7,000 rpm). The use of porous media to represent the bearings in the CFD simulation gave good results on the calculation of the pressure drop through the bearings [35]. This finding may be used in future simulations where the effect of the air flow through the bearings is considered. As this is beyond the scope of this work, these results have been used only for qualitative comparison.

2.2.5 Off-takes Design and Scavenge Efficiency

As the design of the scavenge off-take was found to deeply affect the oil distribution [21] and residence volume [29], experiments were performed both at the University of Nottingham and at the KIT to characterize the effect of the sump design on bearing chamber flows. Using the University of Nottingham bearing chamber test rig, Chandra [24] performed a parametric study of the Rolls-Royce AE3007 sump geometry. The latter is a shallow sump, designed to minimise the oil residence time inside the chamber. The experimental findings indicate that the oil inlet conditions are the most influential parameter on the scavenge performance, while strong secondary air vortices increase the oil residence time, hence they should be avoided. The work gives also an insight on the effect of inner structures such as the “buffer ring”, a bearing support device that tends to prevent the development of strong secondary air flows, reducing the oil residence time. The considerations presented by Chandra [24] are particularly useful when implementing the simulation of a real bearing chamber geometry, as they provide a qualitative insight on the effect of inner structures on the air and oil flows.

Other experimental results for shallow and tangential off-takes are presented by Simmons and Chandra [30], and Chandra et al. [25]. Based on the experience acquired on sumps, the latest experiments of Chandra and Simmons [26] led to the proposal of novel shallow designs, developed to reduce the oil residence volume.

While the work performed on bearing chamber sumps at Nottingham focused mainly on oil residence volume and residence times, investigations carried out at the KIT by Kurz

et al. [10, 21] addressed the bearing chamber scavenge efficiency and oil distribution, assessing the effect of both the sump and vent hole design. The experiments in [10] cover three different scavenge configurations (baseline with rounded hole, shallow ramp, deep ramp with cover) together with nine different vent hole configurations. The results indicate the covered ramp design as the one giving the highest scavenge efficiency, with values between 77% and 97% at 15,000 rpm against the 21–56% of the baseline design. Kurz et al. [10] found also that a protruding vent configuration gives a significant increase in scavenge efficiency, as it prevents the oil film from being ingested by the vent port. As will be illustrated in Chapter 5, the Rolls-Royce bearing chamber simulated in this thesis work is a good example of design following the above guidelines.

2.3 Numerical Modelling

2.3.1 Overview

Among the numerical approaches available to model two-phase flows, the CFD research works on bearing chambers found in the literature rely generally on the following three methods and their variations:

- (Eulerian) Thin Film Model (TFM)
- Volume of Fluid (VOF) method
- Smoothed Particle Hydrodynamics (SPH)

with the work group at Nottingham active on all three. While the TFM and VOF method rely on the Eulerian description of the flow, the SPH approach is a Lagrangian method. The difference between these two approaches lies in the kinematics description adopted:

- Eulerian: the flow field is described through its properties as function of time and position, i.e.

$$\mathbf{U} = f(\mathbf{x}, t) \quad (2.7)$$

where $\mathbf{U}$ is the flow velocity vector, $\mathbf{x}$ the position in the flow field and t the time.

- Lagrangian: the flow is followed in its path, thus it is considered as a set of discrete fluid parcels, identified by their position at any given time:

$$\mathbf{x} = f(\mathbf{X}, t) \quad (2.8)$$

where $\mathbf{X}$ is the initial position of the considered fluid parcel.

The suitability of all three methods is bound to their capability to model, with an acceptable computational cost, flow conditions found typically in bearing chambers, such as shear-driven liquids or droplet and ligament formation. As it will be discussed in the following sections, the TFM gives generally a faster solution than VOF, but is not as accurate when thick films are present; analogously, the SPH has its main strength in being a meshless method, but has still limited capabilities and only recently started being available in commercial software.

Over the years, variations of the three methods described above have been introduced to improve their accuracy to capture some characteristics of bearing chamber flows. These modifications include turbulence damping in VOF [39-43], the coupled Level-set VOF method [42-44], the DPM-VOF model [45-47], the TFM-VOF approach [48], and adaptive meshing in VOF [34, 44]. In the following sections, these will be referred as “enhanced” models.

As the path to the selection of the numerical approach in this thesis has been defined by evaluating the characteristics of the above methods, the following sections give an overview of their fundamental aspects, as well as providing relevant references. The principle underlying the final selection of the VOF method among the three available has been the evaluation of the advantages and disadvantages in relation to the specific flow conditions to be modelled. The need to simulate a real geometry with state of the art techniques, pushing the borders of their applicability, has been the main driver. With the following sections, the link between the problem and the choice of the numerical method is going to be defined.

2.3.2 Thin Film Model (TFM)

During operation, the bearing chamber outer (or “rim”) wall is typically covered by a thin layer of oil. Experimental measurements [10, 22] show that the average oil thickness on the wall is about 1 mm or less, as illustrated in Figure 2.5. If the numerical solution of the film flow is computed explicitly, the need to resolve a large numerical domain while capturing small features such as a film of the order of 2% of the annular cross-section height can lead to significant mesh requirements. In a similar fashion to boundary layer modelling in the theory of turbulence (reported in Section 3.4.6), the Thin Film Model (TFM) avoids the explicit resolution of liquid films on walls, treating them as a boundary condition instead. The flow-governing equations are depth-averaged through the film, assuming the flow as two-dimensional, i.e. without a radial component of velocity. Velocity

and temperature profiles are implemented consequently. In general, the latter profiles are quadratic [8, 49, 50], although in the recent work of Kakimpa et al. [51] orders up to the fourth have been used in the velocity profile to model flow recirculation and improve heat transfer capture.

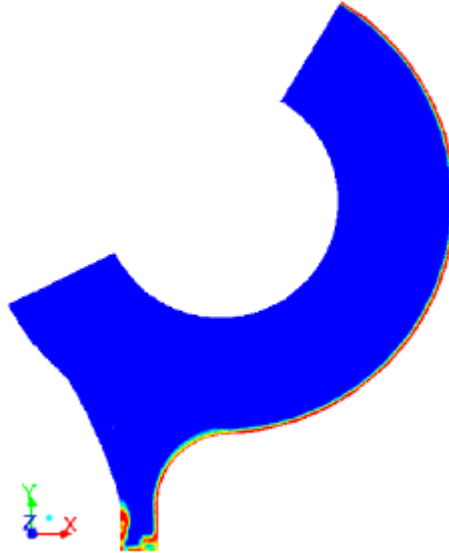


Figure 2.5 – Typical film thickness in a bearing chamber section [43]

Because of the advantage in terms of computational power demand over VOF and SPH, the earliest bearing chamber models found in the literature relied generally on the TFM approach. The first application for bearing chambers, developed at the University of Nottingham Transmissions UTC, is the model of Farrall [49]. His approach was designed to compute the rimming flow in a smooth cylinder and included droplet to film interaction, although several assumptions about the latter were introduced due to the lack of experimental data [49]. Modelling work on thin film flows over multiple obstacles, namely hemispheres and cylinders, was successively done by Baxter [52], again at Nottingham, who evaluated their effect on the oil film flow. In this case, the flow was only gravity driven, thus not considering the air shear effects, and the geometry considered was a flat plate. The work proved that narrowly spaced obstacles, i.e. with a spacing of the order of their diameter, behave as a large obstruction. The latter finding is of particular interest for bearing chamber design, as conditions for oil dry-out in presence of realistic obstacles are identified [52].

Improvements to the thin film model developed at Nottingham were introduced by Williams [53], who enhanced the droplet impact model, analysing the instability effects due to the momentum transferred to the film. Flow recirculation within the film and thermal

effects were also taken into account. The liquid film temperature distribution across its thickness was expressed by Williams as function of the Biot number [53], on the assumptions of uniform wall temperature and of a dominant effect of conduction over convection in the film. The latter hypothesis holds only for thin films, thus the model covers only one of the bearing chamber flow regimes, as described in Section 2.2.3. This problem was later addressed by Kay et al. [50], who extended the thermal formulation of the thin film model to cover the film pooling condition and the transition from smooth film to the latter, for a constant Biot number. The bearing chamber modelling in the work of Williams [53] included also a mass removal region on a defined section of the outer rim, namely a mass sink. It was found that the position of the sink affects the presence of a recirculation area in the film, which is minimized when the sink is placed under it. This observation highlights the importance of the sump location to avoid oil recirculation.

The investigations reported above helped better understand the oil behaviour as a thin film, and led to the development of several models addressing separate bearing chamber flow aspects, such as oil recirculation or droplet-film interaction. Integration work was carried out successively by Wang and Morvan [54]. The authors also attempted a simulation of the KIT bearing chamber (described in Section 2.2), but did not get to conclusive results. This was due to the complexity of the test case and a first CFD version of the thin film model that lacked completeness in the description of the physics involved [54].

The work in [54] highlighted the limitations of the Nottingham thin film model, at this stage, on covering all the typologies of flow found in bearing chambers. A major obstacle was the build-up of thick films at certain locations of the annulus. In these locations, the model (as it was formulated then) was no longer suitable and generally led to non-realistic solutions [7, 55]. To identify the cases in which oil tends to build-up, Kay et al. [50] proposed a parameter λ , named gravity parameter, which describes the conditions for which the film is shear driven, gravity driven, or transitioning between the two states. The gravity parameter depends on a reference film height, reference film velocity, and on the volume fraction of liquid filling the bearing chamber. Despite the better understanding of the switching mechanism between thin and thick film solutions, the challenges of the TFM on the transition case remained. Only recently the work led by

Kakimpa et al. [51] has successfully addressed the various shortcomings of the model, from additional stabilising forces to higher order numerical treatments.²

To include the modelling of droplets in the bearing chamber flow, Jingyu et al. [31] proposed a coupled Discrete Phase Model (DPM)-TFM approach including splashing and oil stripping. The thin film model in this work, however, relies still on quadratic film velocity profiles [31], implying that only thin films without transition to thick can be described appropriately.

Considering the assumption of negligible film thickness compared to the size of the domain, the thin film model is not suitable in general for thick recirculation zones, where a considerable portion of the bearing chamber can fill-up with oil. For this reason, its pure implementation to cover the totality of bearing chamber flow cases is unlikely to give accurate results. Research led by Prof Morvan and Dr Kakimpa at the University of Nottingham's G2TRC has resulted in a combined Thin Film–Volume of Fluid model, covering thin and thick film applications benefitting from the strengths of both methods. As the development of this approach has been completed at the time of the writing of this thesis, its deployment on cases that are more complex than the KIT bearing chamber geometry still requires further testing.

2.3.3 Smoothed Particle Hydrodynamics (SPH)

The Smoothed Particle Hydrodynamics method is a Lagrangian approach developed in the 1970s to investigate astrophysics problems [56] through a “gas dynamics” model. The method uses statistical techniques to match the analytical expressions of the fluid physical variables, as described by Gingold and Monaghan [57] in their first application to simulate non-spherical stars evolution. In generic SPH applications, the fluid is modelled as a set of discrete particles, characterized by inter-particle forces to simulate pressure and the other forces acting in the fluid, such as viscosity and surface tension. One of the advantages of this method is that, unlike the Eulerian methods, mass conservation is always satisfied. Moreover, as it is a meshless method, SPH does not require work to define a suitable numerical grid prior to the simulation. The interface between the two phases is automatically defined by the position of the particles; hence, dedicated methods such as interface tracking or level sets are not required [58].

² The journey on the development of the model at Nottingham has been the object of a special lecture given by Prof Morvan in October 2016, and available online at <https://www.youtube.com/watch?v=e8RMFwKNeF8>

The main challenges in the application of the SPH method to bearing chambers are related to the modelling of liquids, walls and other engineering features such as inlets, outlets, and pressure boundaries. The interaction of fluids having very different densities, namely air and oil, is also particularly challenging to model in SPH: this approach was originally designed for single-phase fluid modelling, while in the bearing chamber problem the interactions between the two phases are a fundamental component of the flow. The problem was addressed at Nottingham by Kruisbrink and Morvan [59], through the development of a pressure boundary concept to represent the pressure exerted by the chamber air. Successively, the air-oil shear forces at the interface and the effect of droplet impacts were included [60].

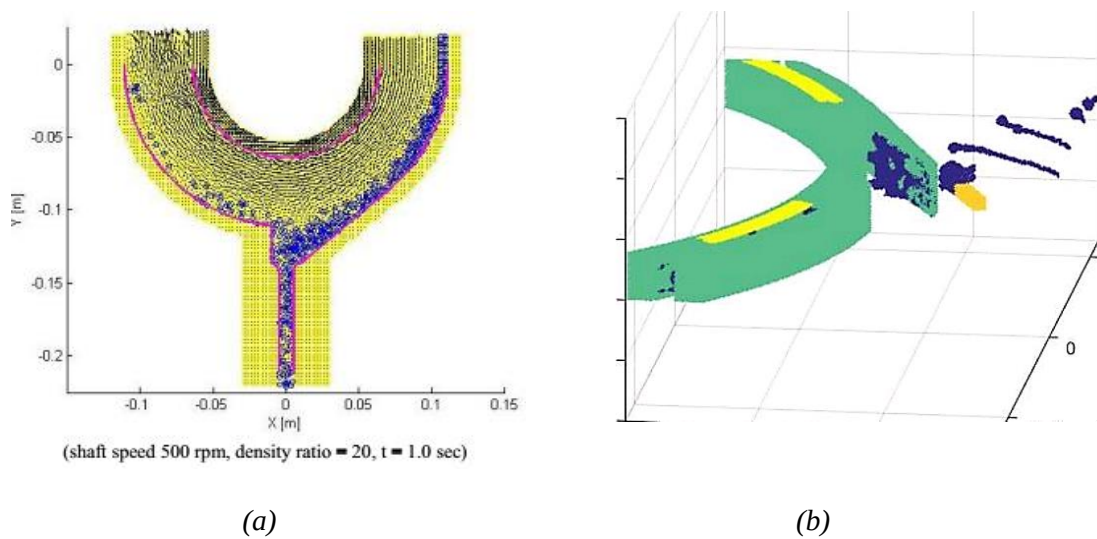


Figure 2.6 – SPH simulations of aero engine oil systems : (a) Two-phase, 2D KIT bearing chamber [60]; (b) Single-phase, 3D scoop [61]

Initial limitations in bearing chamber modelling with SPH were given by the density ratios between the two phases, with the oil to air density ratio limited to 100 for realistic solutions [60], and a goal of 800. Subsequent work led to successful testing of density ratios up to 1000 [62], thus overcoming the previous limitation. Critical aspects of the method are still the creation of complex 3D geometries [61] and the resolution of the liquid phase needed to model thin films and droplet break-up [63]. The latter aspect depends on the number of particles adopted, as illustrated in Figure 2.6. Higher resolutions require large numbers of particles, which can make the computational requirements challenging even on a parallelised code. An example of this aspect is the attempt to simulate the KIT bearing chamber with SPH performed by Wieth et al. [63], which was done using a two-dimensional geometry due to computational power constraints. Despite the promising

outcome [63], the 2D formulation adopted for a highly three-dimensional flow limited the significance of the results for validation.

Considering the progress made in the last three years [60-63], the SPH method appears in fast evolution towards its applicability to three-dimensional bearing chambers. For the time being, additional validation is needed, in particular on three-dimensional cases representative of bearing chamber flows. Moreover, the effectiveness of the code depends on its efficient parallelisation, which is currently being carried out at the University of Nottingham by the research group led by Dr Kruisbrink. Although the applicability of SPH for the simulation of complex bearing chamber geometries is still premature, it is foreseen that in the following years the method will likely become a very powerful resource for applications of this kind.

2.3.4 Volume of Fluid (VOF) Method

Background

The Volume of Fluid (VOF) method is a Eulerian approach developed in the late 1970s for multiphase flow modelling. Its first formulation appeared in the work of Hirt and Nichols [64]. In the VOF method, the fluid is treated as a mixture of multiple phases. The physical properties of the mixture are the average of those of the composing phases, proportionally to the volume of each phase filling the given cell. The quantity α is defined as phase volume fraction, and has a range between 0 and 1. In the case of two phases, the extremes $\alpha = 0$ and $\alpha = 1$ indicate that the given cell is completely filled by one of them. Conventionally, the location of the interface in this case is set at $\alpha = 0.5$ [58].

Due to its expression in Eulerian coordinates, the volume fraction function is both space and time dependent. In a two-dimensional case, the variations of α are governed by

$$\frac{\partial \alpha}{\partial t} + u \frac{\partial \alpha}{\partial x} + v \frac{\partial \alpha}{\partial y} = 0 \quad (2.9)$$

where u and v are the x- and y-components of the flow velocity. Generalising for a 3D case, Equation (2.9) becomes

$$\frac{\partial \alpha}{\partial t} + \mathbf{U} \cdot \nabla \alpha = 0 \quad (2.10)$$

Interface Computation

Considering a cell containing the interface boundary in a 2D case, once α and its gradient are determined the cell is cut through a line orthogonal to $\nabla\alpha$, respecting the volume fraction inside it. The line approximates the interface between the two phases, and is typically discontinuous between the adjacent cells [64]. A popular algorithm for the interface reconstruction is the Piecewise Linear Interface Construction (PLIC). This method aims at maintaining a nearly-smooth interface by using segments for its reconstruction, based on the gradient of the volume fraction in an approach similar to that presented in [64]. A representation of the surface reconstruction with the PLIC method is illustrated in Figure 2.7(d).

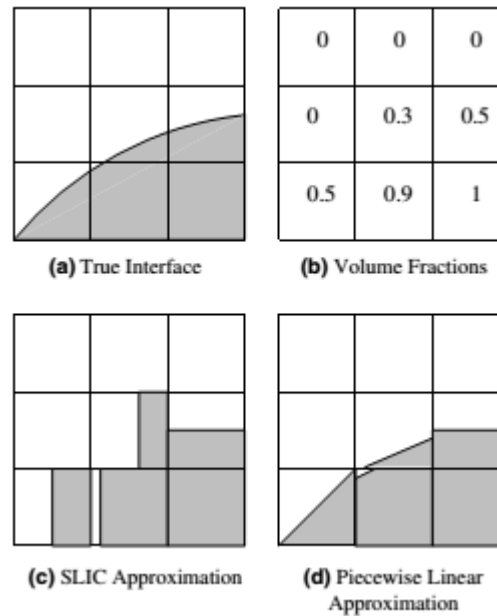


Figure 2.7 – Illustration of surface reconstruction approaches for the VOF method [65]

Other approaches to capture the interface include front tracking methods [66, 67], which use a moving mesh to resolve the interface. Among the advantages, front tracking overcomes the need for surface reconstruction, while improving the estimation of surface-related forces. However, in case of strong deformations of the interface, the mesh regeneration may become excessively costly or lead to failure [58]. An alternative solution is to couple the VOF method with the Level-Set function (later discussed in Section 3.3.4), relying on the Coupled Level Set-VOF method (CLSVOF) [68, 69]. This approach aims at combining the advantages of the two methods, giving a smooth interface between the phases thanks to the Level-Set function while preventing mass loss using the VOF equations. Although this method can lead to very accurate results, in particular for surface

tension-dominated flows [69], the need to resolve an additional equation for the Level-Set function implies higher computational demands compared to the pure VOF approach. Thus, the use of the CLSVOF is likely to be non-viable for the simulation of long transients in a full-scale bearing chamber at this stage.

A recent approach becoming widely spread in gas-liquid simulations is the Compressive interface reconstruction as formulated in ANSYS Fluent [70, 71]. Considering the applications of interest for this research work, this method showed promising results in droplet splashing simulations [9] as well as on bearing chamber applications [34]. Together with the PLIC Geometric-Reconstruction approach, it is one of the most accurate methods for interface computation available in the pure VOF method [71], and brings the benefit of a lower computational requirement [71]. Therefore, as opposed to the previously discussed front tracking and CLSVOF, it is a good candidate for the applications required in this thesis. Its discussion is reported in Section 3.3.4.

Stratified Flows

Stratified flows are characterised by a vertical distribution of fluids having different densities. The fluids tend to form layers flowing parallel to each other, according to the effect of gravity [72]. A common two-phase example is a flow of water and air in a horizontal channel, as illustrated in Figure 2.8.

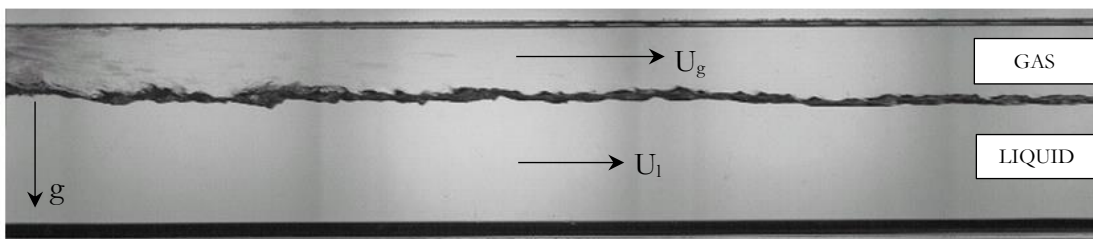


Figure 2.8 – Example of gas-liquid stratified flow (adapted from [73])

The modelling of stratified flows has a strong importance for bearing chamber simulations. On the bearing chamber walls, a stratified flow condition is found typically between oil films and the air flowing above them. The modelling of this region in CFD is challenging, since phenomena having smaller length scales than the grid resolution normally take place at the gas-liquid interface. These can have a large influence on the turbulent kinetic energy of the liquid phase, influencing the accuracy of the prediction [73]. The inability to resolve explicitly this sub-grid component of the momentum exchange calls for the need to model the interaction between the two phases. In the first place, the

latter needs to be understood comprehensively, relying on simple cases such as stratified flows in straight channels. The first to acknowledge the need for turbulence modelling at the gas-liquid interface and to provide a solution attempt were Akai et al. [74] in 1981, followed by other research works [75-79]. These early attempts, however, adopted empirical correlations, dependent on the specific case to be modelled. This limitation was overcome by Egorov [80], who introduced a wall-like turbulence damping at the interface in two-phase (Euler-Euler) models, to control the momentum exchange between gas and liquid. The novelty of Egorov's method [80] is its applicability to a wide range of flow conditions, as well as its independence of local specifications such as the liquid layer thickness and the surface wave properties [80]. One limitation remaining in this approach is the need to select a scaling factor dependent on the mesh size at the gas-liquid interface. This factor should have a value of 10 for a cell y^+ value of 1 at the interface, and should be higher for y^+ values higher than 1 [80], although a correlation is not provided in [80]. A detailed description of Egorov's turbulence damping method is given in Section 3.5.2.

As the turbulence damping approach of Egorov [80] was conceived for nuclear applications, many examples of its implementation are found in this field. The majority of research on this topic adopts Euler-Euler models [73, 81-84], with the only examples of VOF found in the work of Bartosiewicz et al. [85, 86]. In this particular stratified flow application, the use of VOF combined with Egorov's turbulence damping [80] did not lead to satisfactory results compared to the Euler-Euler approach [85]. This prompts some questions on whether VOF is the "right" choice for modelling the stratified flows found in bearing chambers. Moreover, in none of the works mentioned above was found a discussion on the effect of the choice of a different turbulence damping factor. The work described in this thesis aims to provide an answer to both these questions.

Bearing chamber-oriented work on gas-liquid interaction modelling in stratified flows was performed by Hashmi et al. [40] in 2010, and Hashmi [39], who developed two turbulence damping approaches alternative to the one proposed by Egorov [80]. These

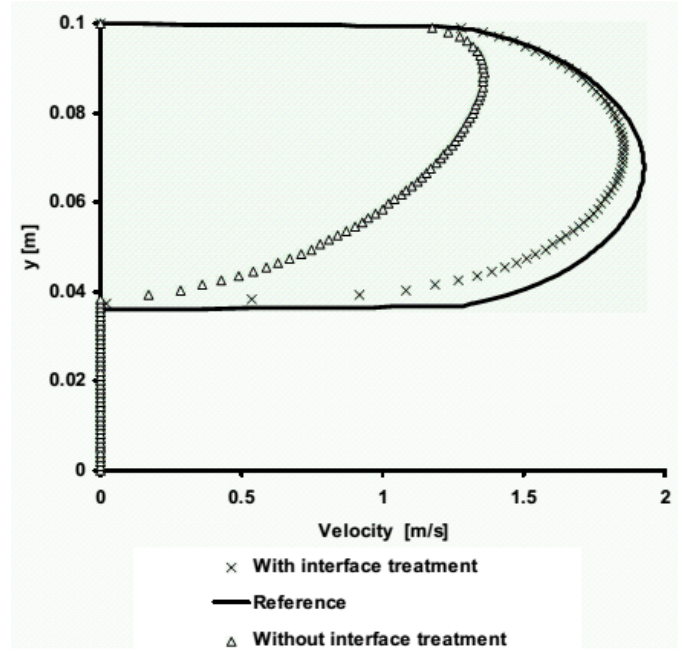


Figure 2.9 – Enhancement of velocity profiles prediction in VOF through gas-liquid interface treatment [40]

were implemented in VOF and demonstrated their suitability in terms of accuracy of velocity profiles predictions in stratified flows, as illustrated in Figure 2.9. Both methods are described in Section 3.5.3.

While Hashmi et al. [40] focused on the development of novel turbulence damping methods, research activities conducted at the University of Nottingham by Tkaczyk and Morvan [43] in the SILOET project led to the implementation and testing of Egorov's approach [80] with VOF for both k - ω and k - ε turbulence models. One major achievement in [43] was the definition of an optimal value for the scaling factor of the turbulence damping function [80], based on the matching of velocity and turbulence kinetic energy profiles with two reference stratified flow experiments [87, 88]. Despite the recommended turbulence damping factor value of 100 [43], uncertainties on the suitability of this value emerged when implementing it on bearing chamber models [89, 90]. Centrifugal and gravity forces act on the film in the latter case, making the applicability of the approach far less certain than in the channel flow. The outcome highlighted the need for a better understanding of the impact of the turbulence damping factor on bearing chamber flow conditions, in particular concerning the oil distribution and residence volume [90].

VOF Method in Bearing Chamber Modelling

In the first paper evaluating the applicability of VOF to aero-engine oil systems, Young and Chew [91] identified in bearing chambers a component having flow features

not appropriately treated by this method. Functions capable of better capturing the air-oil interaction, as described in the previous subsection, significantly contributed to overcome the challenges originally encountered [91]. An early example of VOF application to bearing chambers using a dedicated gas-liquid interface treatment is found in the work of Peduto et al. [41], who evaluated its effect on the oil film thickness and on the characteristics of the near-sump recirculation zone against a non-treated baseline case. Although [41] highlights the beneficial contribution of interface treatment on the film thickness by comparing the latter with experimental data, the applicability of the results to a broader case is limited by the boundary conditions adopted. In fact, the imposition of the film thickness value at the boundaries of the section tested, as done in [41], is not possible for a whole bearing chamber model. Nonetheless, the work shows a significant discrepancy of the non-treated case against the treated case, as illustrated in Figure 2.10.

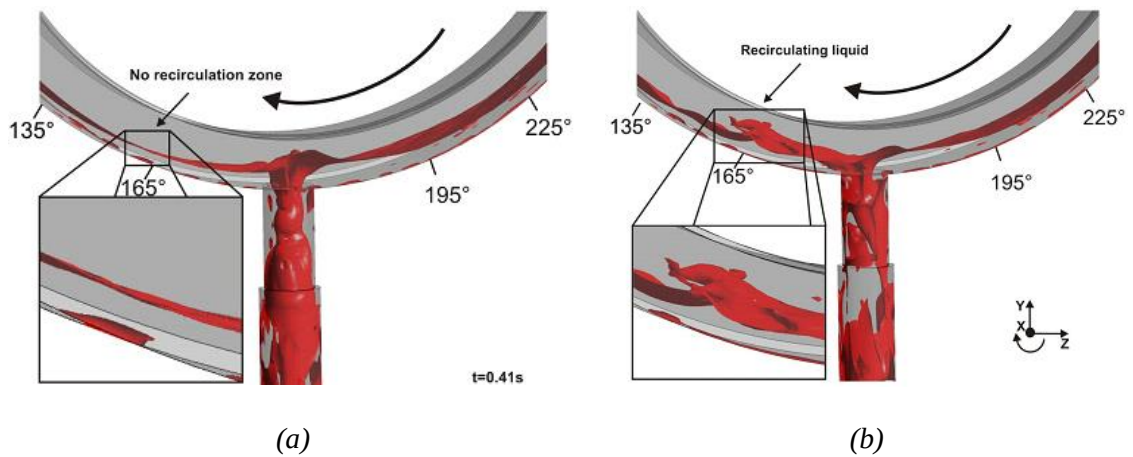
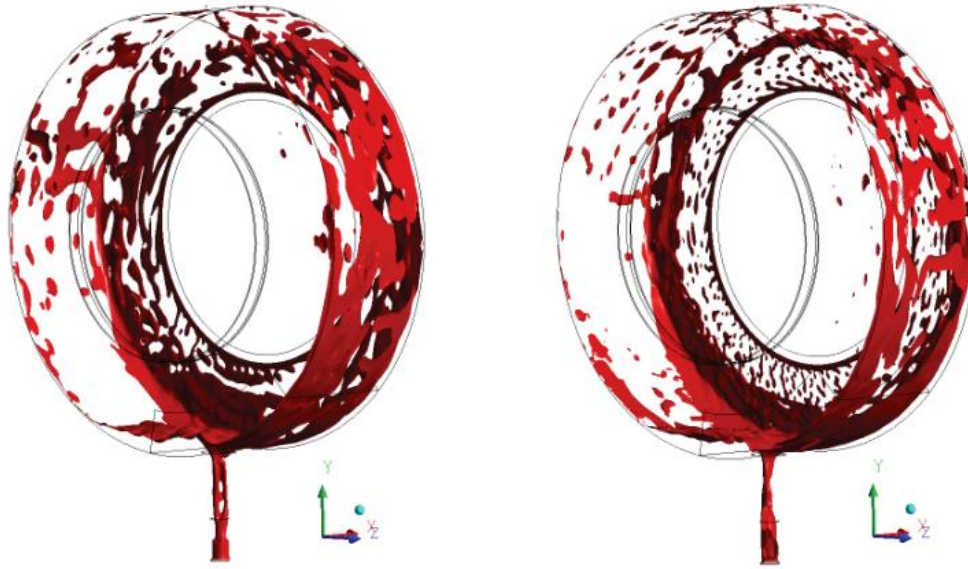


Figure 2.10 – Effect of gas-liquid interface treatment on oil recirculation in bearing chamber section [41]. (a) No interface treatment; (b) Interface treatment

While the early simulations of bearing chambers with VOF were limited to models of sumps [41, 43] because of the significant number of cells required for the full annulus, a whole bearing chamber model was developed by Krug [92] to validate the results against film thickness and scavenge efficiency experimental data [10, 21]. The work of Krug [92] represents a first attempt to model a full bearing chamber with VOF while following a systematic validation approach. Among the major achievements, two different turbulence damping methods, namely Hashmi's simplified interface treatment (SIT) [39] and Egorov's turbulence damping [80], were compared against each other and with experimental data at both low (5,000 rpm) and high (15,000 rpm) shaft speeds.

The results show a tendency of the models to under-predict the circumferential film thickness distribution in the low speed case, and to over-predict it at high speed. While no



(a) Coarse mesh

(b) Fine mesh

Figure 2.11 – Effect of mesh refinement on oil distribution , iso-surfaces of oil volume fraction $\alpha_o = 0.3$ [92]. (a) 621,500 cells; (b) 1,700,000 cells

substantial difference is found between the two turbulence damping approaches at low speed, the results at high speed tend to favour the SIT [92]. Krug [92] found also that the mesh size plays an important role in the accuracy of the results. A preliminary mesh independence study with 621,500 cells and 1.7 million cells highlighted a tendency for oil to form thinner films in the fine mesh case, as well as to break-up into smaller droplets compared to the coarse mesh, as shown in Figure 2.11.

Both the coarse and fine meshes showed inaccuracies in reproducing the continuous film on the chamber wall observed experimentally [92], as well as a relatively poor approximation of the experimental data. Mesh independence could not be achieved, and the discrepancies were attributed to the momentum transfer modelling between phases, as well as to an inaccurate computation of the surface tension [92]. The limited computational power available led Krug [92] to opt for the coarse mesh to perform the remaining evaluations of the VOF method. This implied that a high resolution of the gas-liquid interface could not be achieved, thus limiting the reliability of the results. In particular, the accuracy of the results obtained with Egorov's turbulence damping [80] may have been affected, as Krug [92] adopted the default damping factor value of 10 despite the latter being suitable for a fine mesh at the interface (see Section 3.5.2).



Figure 2.12 – Bearing chamber simulation using adaptive meshing , iso-surface of oil volume fraction $\alpha_o = 0.25$ [34]

In order to improve the resolution of the gas-liquid interface and therefore the accuracy of momentum transfer modelling, Crouchez-Pillot et al. [44, 93] at the University of Nottingham considered adaptive mesh refinement as a possible solution. The work modelled the same bearing chamber topology analysed by Krug [92], and led to promising results with the identification of small oil droplets being released by the bearings. However, the significant computational power required and the limited duration of the project did not allow to achieve the flow stationary state, with only few hundredths of a second simulated. The work of Crouchez-Pillot et al. [44, 93] was then followed up by Krug et al. [34] at KIT, who performed a longer simulation using adaptive meshing and achieved a uniform coverage in oil film of the bearing-side wall of the chamber.

Comparing the oil distribution of with those of Figure 2.11, a clear improvement of the interface resolution is visible. The price to pay for adaptive mesh refinement, however, is a significant increase in computational costs. The mesh size of the latter case ramped up to 7,000,000 cells from the baseline 621,500 [34], which corresponds to an 11.3 times increase. The cell size reduction led to decrease the time step size by a factor of 4.3 to $3.25 \cdot 10^{-6}$ seconds [34]. Combining these two factors with the mesh adaption process, the computational power was found to increase from 1,500 CPUh/s to 63,000 CPUh/s [34], corresponding to a factor 42. Moreover, the work of Krug et al. [34] provides only a qualitative description of the results obtained with adaptive meshing, suggesting that the flow stationary state had not been achieved. Following the challenges encountered with

adaptive meshing, the successive work at KIT using VOF [23] relied on a fixed, 5.9 million cells mesh modelling a 60° bearing chamber sector. Although the latter example focuses on a buffer chamber, thus being not of direct interest for the application in this thesis, the modelling strategy followed confirms that the use of VOF with adaptive meshing is still a challenging option for bearing chambers.

Additional simulation work on the KIT bearing chamber [19] was performed by Yaguo et al. [94] who used the coupled level-set VOF method to evaluate the scavenge efficiency behaviour of different geometries. In this work, however, very little detail is given on the achievement of the flow stationary state. Moreover, no comparison with the reference experimental data [19] is provided.

A different route towards an “enhanced” VOF approach for bearing chambers is the coupled DPM-VOF method, developed by Tkaczyk and Morvan [43] with the aim to contain the mesh size by not resolving explicitly the oil droplets. The DPM-VOF is a hybrid Lagrangian-Eulerian approach, based on the assumption that oil is released from the shaft as small droplets, in the range of 50 to 1500 μm [45]. The droplets are treated as point particles with given mass and velocity, while the oil film on the wall is described with the VOF method. Once the droplets hit the wall or the overlying film, their mass and momentum are converted into VOF by means of a source term. The method was further developed at Nottingham by Adeniyi et al. [45-47], who introduced the effect of droplet splashing with the generation of secondary droplets, as illustrated in Figure 2.13. The effects of air drag on the DPM particles, as well as air-droplet convective heat transfer, were also included [45-47].

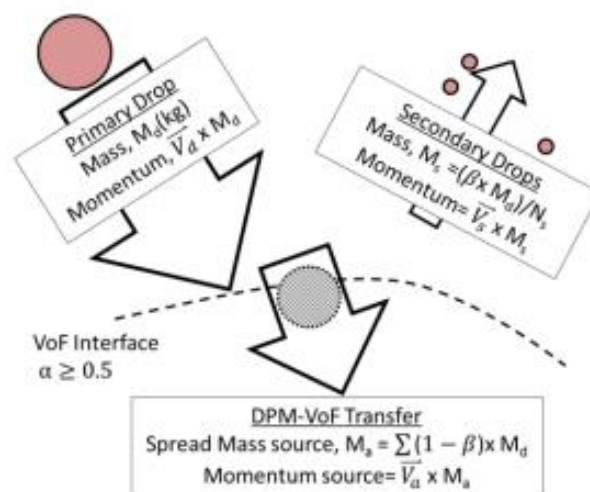


Figure 2.13 – Schematic of DPM-VOF approach [46]

In the work of Adeniyi et al. [45-47], the outcome of the droplet-wall interaction is based on the results of Farrall et al. [95]. Adeniyi et al. [45-47] extensively tested the DPM-VOF method with coupled level-set on bearing chamber flows, simulating the experimental setup developed by Chandra et al. [24] (see Section 2.2.5). The simulations replicated the high shaft speed flows, namely 10,000 and 15,000 rpm, with a mesh size of about 1 million nodes. An example of the resulting oil film formation from Lagrangian particles, and of the heat transfer coefficient distributions obtained, is reported in Figure 2.14.

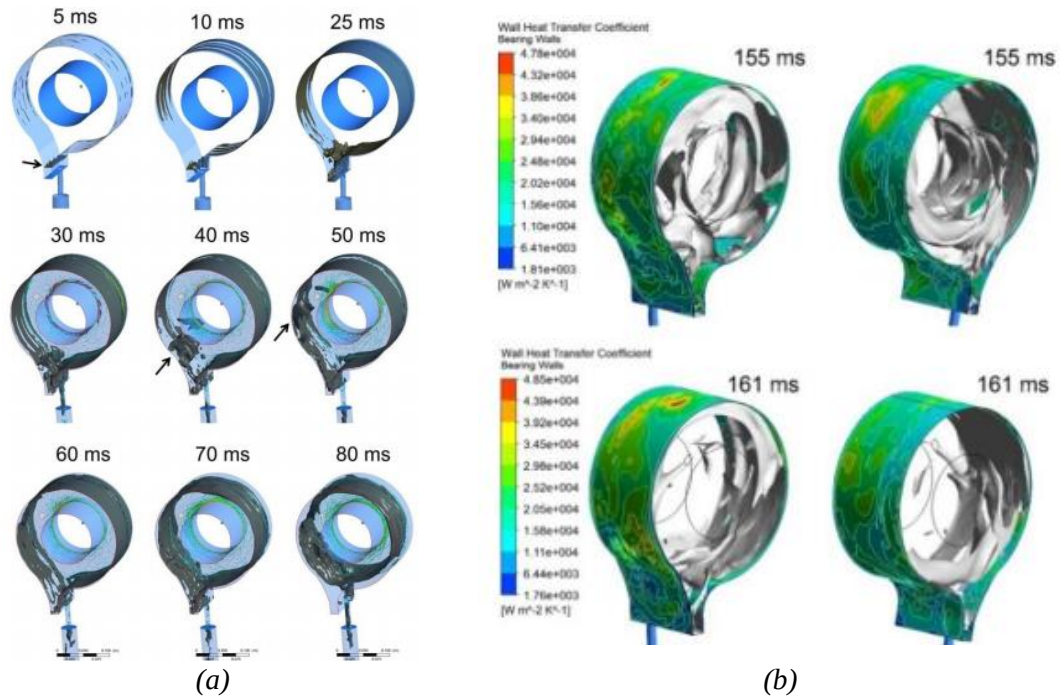


Figure 2.14 – Results with DPM-VOF approach [46]. (a) Film formation from DPM particles, $N=10,000$ rpm, iso-surfaces of $\alpha_o = 0.2$; (b) Wall heat transfer coefficients distribution

In the results of Adeniyi et al. [45-47], the film thickness in the sump region shows good agreement with the experimental data reported in [27], while the average film thickness distribution was overestimated by the model. The cause was identified in the boundary conditions for the rotating inlet distributor [45], highlighting one major weakness of the DPM-VOF application to bearing chambers, namely the uncertainty on the oil release mechanism from the bearings. Investigations were conducted by Adeniyi et al. [96, 97], but a comprehensive understanding of the modes through which oil leaves the bearings, including experimental data, is yet to be achieved.

A potential improvement to the VOF computational speed and accuracy in the near future is likely to come from the Thin Film Model (TFM)-VOF coupling, developed at the

University of Nottingham by Prof Morvan and Dr Kakimpa. As discussed in Section 2.3.2, this approach introduces a transition between thin film modelling and VOF according to the wall film thickness, as described in Figure 2.15. The potential benefit is a significant reduction of the mesh size due to the non-explicit modelling of the thin film regions, together with an improved accuracy due to the use of the two models to capture the flow conditions they have been designed for. As illustrated in Figure 2.16, the first implementation of the concept on a bearing chamber has been successful, although challenges related with the transition from VOF to thin film are still being addressed at the time of the writing of this thesis [48]. Whilst the model is not ready yet for industrial applications, it is likely to become a useful tool in this context in the near future, and it will build, rapidly, on the work proposed in this thesis.

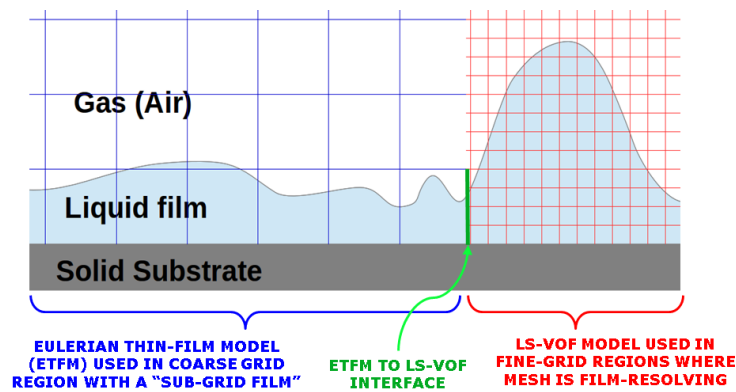


Figure 2.15 – TFM-VOF concept [6]

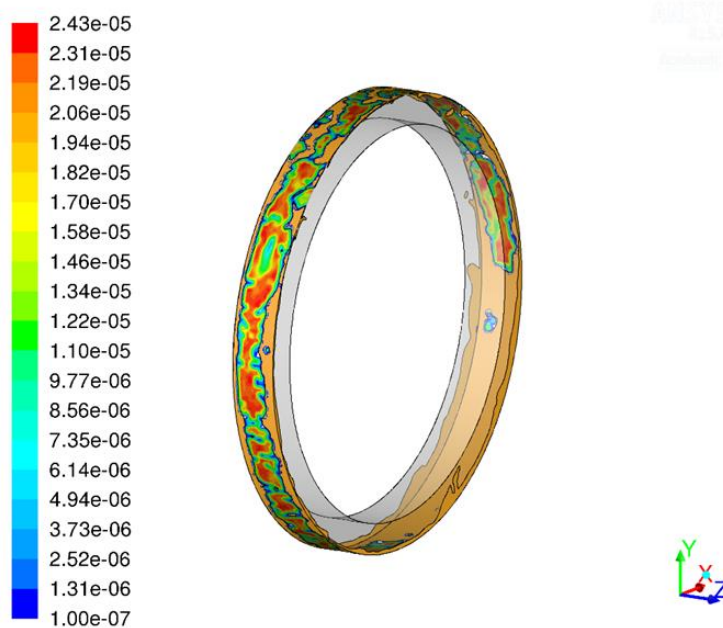


Figure 2.16 – Application of TFM-VOF on bearing chamber . Contours of oil film thickness (TFM) and air-oil interface (VOF) [6]

3 Proposed VOF Modelling Approach

3.1 Introduction

The aim of this chapter is to outline the relevant equations and modelling techniques adopted in this work, in order to provide a solid reference to the results obtained with their application. Since a major objective of this research is to demonstrate the application of state of the art numerical techniques to a real bearing chamber geometry, most of the methods adopted are available as standard in commercial software.

The main aspects of modelling addressed in this chapter include the flow equations, turbulence modelling, and solution techniques. Particular focus is dedicated to the Volume of Fluid method and turbulence damping, as their formulation has a significant impact on the results obtained in the simulations performed.

3.2 Flow Equations

3.2.1 Overview

The equations that govern the flow of fluids are the mathematical expression of the conservation laws of physics. These can be summarised as follows:

- The mass of each fluid particle is conserved
- The rate of change of momentum on a fluid particle is equal to the sum of forces acting on it (second law of Newton), also known as Navier-Stokes Equations
- The change of energy of a fluid particle is equal to the sum of the heat flux and work done on it (first principle of thermodynamics)

The expression of the above laws in mathematical terms gives what are known as continuity, momentum, and energy equations. In the following subsections, each of the above equations is reported for a continuum fluid. The equations refer to a small element of fluid having sides δx , δy , δz .

A comprehensive overview of the flow equations can be found in the work of Versteeg and Malalasekera [98].

3.2.2 Continuity Equation

The continuity equation states that, for a given control volume, the rate of variation of mass within the volume is equal to the net rate of mass flow. This is expressed as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (3.1)$$

where ρ is the fluid density, and $\mathbf{U}$ the flow velocity vector. The first term on the left hand side of Equation (2.1) is the time derivative of the density within the control volume, while the second term is the net flow of mass leaving the volume. As the fluid is assumed incompressible for the flow conditions of our simulations, Equation (2.1) becomes

$$\nabla \cdot \mathbf{U} = 0 \quad (3.2)$$

or, using the long notation, in a Cartesian frame of reference,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (3.3)$$

where u, v, w are the components of $\mathbf{U}$ respectively in the x, y, z directions.

3.2.3 Momentum Equation

In an inertial reference frame and for an incompressible flow, the conservation of momentum is described by the following form of the Navier-Stokes equation

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) \\ = -\nabla p + \nabla \cdot \mu [\nabla \mathbf{U} + (\nabla \mathbf{U})^T] + \rho \mathbf{g} - \mathbf{F}_s + \mathbf{S}_m \end{aligned} \quad (3.4)$$

In (3.4) the inertial terms on the left hand side, respectively unsteady and convective acceleration per unit volume, are balanced by the pressure and viscous stress terms $-\nabla p$ and $\nabla \cdot \mu [\nabla \mathbf{U} + (\nabla \mathbf{U})^T]$, the body forces $\rho \mathbf{g}$ and $-\mathbf{F}_s$, and the source term $\mathbf{S}_m$ which accounts for the effect of additional body forces. The term $\mathbf{F}_s$ represents the surface tension force, which is computed at the free surface for multiphase flows.

3.2.4 Energy Equation

The energy conservation equation implemented in the VOF model, which relies on the pressure-based solver (see Section 3.6.3), has the following form

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot [\mathbf{U}(\rho E + p)] = \nabla \cdot (k_{eff} \nabla T) + S_h \quad (3.5)$$

In (3.6), E is the internal energy of the given fluid particle, and k_{eff} the effective thermal conductivity, defined as

$$k_{eff} = k + k_t \quad (3.6)$$

where k is the fluid thermal conductivity and k_t its turbulent thermal conductivity, which depends on the turbulence model used [70]. The term S_h is the heat generation source. In our simulations, this term is zero as no heat generation is assumed in the domain.

The VOF model, described in the following section, mass-averages both energy and temperature in the control volume as follows,

$$E = \frac{\sum_{i=1}^n \alpha_i \rho_i E_i}{\sum_{i=1}^n \alpha_i \rho_i} \quad (3.7)$$

$$T = \frac{\sum_{i=1}^n \alpha_i \rho_i T_i}{\sum_{i=1}^n \alpha_i \rho_i} \quad (3.8)$$

In (3.7) and (3.8) i represents the i -th phase, while α is the phase volume fraction.

3.3 The Volume of Fluid (VOF) Model

3.3.1 Overview

The Volume of Fluid (VOF) model is designed to compute the behaviour of two or more immiscible fluids while solving a single set of momentum equations. This is done by averaging the fluid properties, proportionately to the volume fraction of the phases filling each control volume, and sharing the same velocity for all phases. According to this definition, the density and viscosity required by the momentum equation (3.4) in the control volume are expressed as

$$\rho = \sum_{i=1}^n \alpha_i \rho_i \quad (3.9)$$

$$\mu = \sum_{i=1}^n \alpha_i \mu_i \quad (3.10)$$

which in the case of a gas-liquid flow become

$$\rho = \alpha_g \rho_g + \alpha_l \rho_l \quad (3.11)$$

$$\mu = \alpha_g \mu_g + \alpha_l \mu_l \quad (3.12)$$

where the subscript g indicates the gas phase and l the liquid phase.

In order to track the position of the interface, the VOF model solves a continuity equation for the volume fraction of one of the phases (or $N - 1$ phases for a mixture of N phases). This expression of the continuity equation is named typically volume fraction equation, and has the following form,

$$\frac{1}{\rho_i} \left[\frac{\partial}{\partial t} (\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \mathbf{U}) \right] = \frac{1}{\rho_i} (S_{\alpha_i} + \dot{m}_{net}) \quad (3.13)$$

In (3.13) the term S_{α_i} is a mass flow source for the phase i , while $\dot{m}_{net}$ is the net mass transfer per unit volume between phases in case of evaporation or condensation. In our simulation, both these terms are zero as there are no mass sources in the domain and no change of phase is assumed. The volume fraction equation for the N -th phase is computed by setting the following condition,

$$\sum_{i=1}^N \alpha_i = 1 \quad (3.14)$$

which corresponds to imposing mass conservation.

3.3.2 Implicit Formulation of the Volume Fraction Equation

The volume fraction equation (3.13), on the assumption of no mass sources and no phase change, is discretised in its implicit form as follows,

$$\frac{\alpha_i^{n+1} \rho_i^{n+1} - \alpha_i^n \rho_i^n}{\Delta t} V + \sum_F (\rho_i^{n+1} \dot{V}_F^{n+1} \alpha_{i,F}^{n+1}) = 0 \quad (3.15)$$

In (3.15), $n + 1$ and n are respectively the indexes for the current and previous time step, α_i is the cell value of the volume fraction at the given time step, V the cell volume, $\dot{V}_F$ the volume flux through the face, and $\alpha_{i,F}$ the face value of the i -th volume fraction. The fluxes through the faces are interpolated according to the discretisation scheme selected. The relevant volume fraction discretisation schemes are discussed in Section 3.3.4.

3.3.3 Explicit Formulation of the Volume Fraction Equation

The explicit formulation of the volume fraction equation (3.13), on the assumption of no mass sources and no phase change, is the following,

$$\frac{\alpha_i^{n+1}\rho_i^{n+1} - \alpha_i^n\rho_i^n}{\Delta t}V + \sum_F (\rho_i^n \dot{V}_F^n \alpha_{i,F}^n) = 0 \quad (3.16)$$

Differently from the implicit formulation of equation (3.15), in the explicit formulation (3.16) the volume fraction at the current time step α_i^{n+1} is calculated from the quantities of the previous time step, hence no iterative solution of the equation is required. Analogously to the implicit formulation described in Section 3.3.2, the face fluxes are interpolated according to the chosen spatial discretisation scheme, as reported in the following subsection.

3.3.4 Interface Computation

Geometric Reconstruction

The Geometric reconstruction scheme is a commonly used piecewise-linear interface reconstruction approach (see Section 2.3.4). With this scheme, the interface is represented as a linear slope within each cell, and its orientation used to compute the advection of fluid through the faces of the cells.

The Geometric reconstruction scheme is composed of three steps. In the first step, the position of the interface with respect to the centre of the cell is computed from the information about the volume fraction and its derivatives in the given cell. The second step calculates the fluid advection through each face based on the position and orientation of the interface, as well as on the velocity distribution on it. Finally, the volume fraction is computed from the balance of the fluxes obtained in the previous step. A comprehensive discussion of the method, including the equations implemented, can be found in the work of Rider und Kothe [99].

In ANSYS Fluent, the Geometric reconstruction scheme is available only with the VOF explicit formulation (Section 3.3.3). Because of this constraint, the scheme was found particularly heavy to run on bearing chamber simulations, and its use was limited to short test runs.

High Resolution Interface Capturing (HRIC)

The HRIC is a composite scheme consisting of a nonlinear blend of upwind and downwind differencing, in order to get a bounded discretisation free from numerical diffusion for the convective part of the volume fraction conservation equation [100]. In this scheme, proposed originally by Muzaferija et al. [100], the normalised volume fraction in the cell $\tilde{\alpha}_c$ is computed as follows

$$\tilde{\alpha}_c = \frac{\alpha_D - \alpha_U}{\alpha_A - \alpha_U} \quad (3.17)$$

where D , U , A are respectively the donor, upwind, and acceptor cell. The normalised face volume fraction $\tilde{\alpha}_F$ is given by

$$\tilde{\alpha}_F = \begin{cases} \tilde{\alpha}_c & \text{if } \tilde{\alpha}_c < 0 \text{ or } \tilde{\alpha}_c > 1 \\ 2\tilde{\alpha}_c & \text{if } 0 \leq \tilde{\alpha}_c \leq 0.5 \\ 1 & \text{if } 0.5 \leq \tilde{\alpha}_c \leq 1 \end{cases} \quad (3.18)$$

To address the case in which the upwind value is not available, such as in an unstructured mesh, a modification to the HRIC was introduced in [70], as directly using the value of $\tilde{\alpha}_F$ would generate wrinkles if the flow is parallel to the interface. In this version of the scheme, known as modified HRIC [70], the angle θ between the face normal and interface normal is used to compute a corrected face value $\tilde{\alpha}_F^*$ of the volume fraction:

$$\tilde{\alpha}_F^{UQ} = \begin{cases} \tilde{\alpha}_c & \text{if } \tilde{\alpha}_c < 0 \text{ or } \tilde{\alpha}_c > 1 \\ \min\left(\tilde{\alpha}_F, \frac{6\tilde{\alpha}_c + 3}{8}\right) & \text{if } 0 \leq \tilde{\alpha}_c \leq 1 \end{cases} \quad (3.19)$$

$$\tilde{\alpha}_F^* = \tilde{\alpha}_F \sqrt{\cos \theta} + (1 - \sqrt{\cos \theta}) \tilde{\alpha}_F^{UQ} \quad (3.20)$$

where

$$\cos \theta = \frac{\nabla \alpha \cdot \mathbf{d}}{|\nabla \alpha| |\mathbf{d}|} \quad (3.21)$$

with $\mathbf{d}$ the vector that connects the centres of the cells in contact with the face F . The face volume fraction can be then computed as:

$$\alpha_F = \tilde{\alpha}_F^* (\alpha_A - \alpha_U) + \alpha_U \quad (3.22)$$

Compressive Scheme

The Compressive scheme is a second order reconstruction scheme based on the slope limiter. This scheme performs a second order interpolation of the interface, which is previously calculated with a donor-acceptor scheme [9]. The interpolation method is defined with the following expression:

$$\alpha_F = \alpha_D + \beta \nabla \alpha_D \quad (3.23)$$

where α_F is the face value of the volume fraction, α_D is the donor cell value of the volume fraction, $\beta = 2$ is the slope limiter, and $\nabla \alpha_D$ is the donor cell volume fraction gradient.

Coupled Level-Set VOF Method

Although the coupled level-set VOF method has not been used extensively in our simulations, it is reported in this section for completeness as it represents a frequently used option to accurately describe the interface topology in two-phase flows. This is particularly useful where the interface curvature has a relevant effect on the flow behaviour.

The level-set method was introduced by Osher and Sethian [101] as an interface tracking approach based on the advection of a continuous function representing the interface between two phases. The function, called level-set and identified by ϕ , is defined as a given distance from the interface. The value $\phi = 0$ corresponds to the position of the interface, while $\phi > 0$ defines the primary phase and $\phi < 0$ the secondary phase. The advection of ϕ is described by

$$\frac{\partial \phi}{\partial t} + \mathbf{U} \cdot \nabla \phi = 0 \quad (3.24)$$

thus the function, starting from the initial position of the interface, will follow its time-evolution according to the flow velocity field $\mathbf{U}$.

Since the level-set function is smooth and continuous, it enables a good estimation of the interface curvature κ , thus improving the accuracy of the computation of the surface tension force $\mathbf{F}_s$ required in the Navier-Stokes equation (3.4). These two quantities are computed as reported in Equation (3.25) and (3.26).

$$\kappa(\phi) = \nabla \cdot \frac{\nabla \phi}{|\nabla \phi|} \Big|_{\phi=0} \quad (3.25)$$

$$\mathbf{F}_s = \sigma \kappa \delta(\phi) \mathbf{n} \quad (3.26)$$

where the local normal to the interface $\mathbf{n}$ is given by

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|} \Big|_{\phi=0} \quad (3.27)$$

and

$$\delta(\phi) = \begin{cases} 0 & |\phi| \geq a \\ \frac{1 + \cos\left(\frac{\pi\phi}{a}\right)}{2a} & |\phi| < a \end{cases} \quad (3.28)$$

where a is the interface thickness.

Since the formulation of the level-set function has no built-in volume conservation, mass can be gained or lost at each time step, leading to the accumulation of errors [102].

For this reason, the coupled level-set and VOF (CLSVOF) method aims at combining the interface reconstruction accuracy of the level-set and the mass conservation properties of the VOF method. In the CLSVOF formulation of Sussman and Puckett [69], density and viscosity are derived from the level-set function as follows

$$\rho(\phi) = \rho_g(1 - H(\phi)) + \rho_l H(\phi) \quad (3.29)$$

$$\mu(\phi) = \mu_g(1 - H(\phi)) + \mu_l H(\phi) \quad (3.30)$$

where $H(\phi)$ is the smoothed Heaviside function defined by

$$H(\phi) = \begin{cases} 0 & \text{if } |\phi| > a \text{ in gas phase} \\ 1 & \text{if } |\phi| > a \text{ in liquid phase} \\ \frac{1}{2} \left[1 + \frac{\phi}{a} + \frac{1}{\pi} \sin\left(\frac{\pi\phi}{a}\right) \right] & \text{if } |\phi| \leq a \end{cases} \quad (3.31)$$

The level-set function is then re-initialised at each time step in order to avoid the growth of errors in the mass and momentum solutions [70]. Both the VOF and level-set function values are used to compute the position of the interface through piecewise linear interface construction (PLIC). The algorithm of the implementation of the CLSVOF method is reported in Figure 3.1.

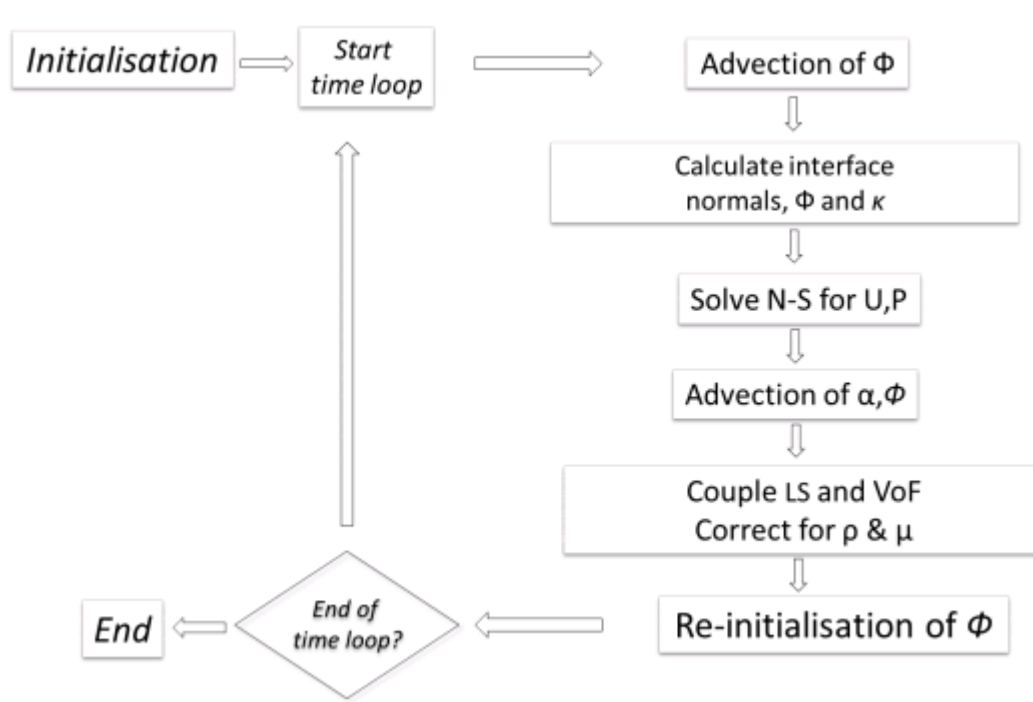


Figure 3.1 – Implementation of coupled level-set VOF method [45]

Although the CLSVOF method enhances the accuracy of the interface reconstruction as described above, its implementation corresponds to one additional advection equation to be solved in the computation loop. Considering the objective of running the bearing

chamber simulations fast enough to achieve the flow stationary state in relatively short computational times, it was decided to trade-in some level of detail in order to make the simulations affordable. In fact, early tests highlighted that an increase in simulation speed and stability could be achieved when the level-set function is not used. Moreover, as the CLSVOF method in ANSYS Fluent requires the mesh to be either hexahedral, tetrahedral, or a combination of both for 3D cases [70], its implementation on the BR725 Rear Bearing Chamber simulation of Chapter 5 would have limited the mesh choices. This would have led to miss the advantages of the polyhedral grid (see Section 5.2.2 – Mesh), posing significant size and quality challenges on an alternative mesh.

Other schemes adopted for the interface computation in VOF include the Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM) introduced by Ubbink and Issa [103]. Analogously to the Geometric reconstruction, this scheme is limited by its first order formulation [104], which makes it available only for the explicit formulation of the VOF method [70]. Therefore, it has not been investigated further, as according with the observations made in Chapter 2 the latter formulation leads to time step size requirements which cannot be afforded in the full bearing chamber simulation.

3.4 Turbulence Modelling

3.4.1 The Reynolds-Averaged Navier-Stokes (RANS) Equations

In turbulent flows, flow fields such as velocity and pressure vary with time in a random manner, thus a computation fully describing the motion of all the fluid particles is not practically viable. An approach generally adopted is to express the flow variables as the sum of their mean value and the statistical value of their fluctuation [98]. For example, the velocity field will be defined by $U(t) = \bar{U} + u'(t)$, where $\bar{U}$ is the mean flow velocity and $u'(t)$ the oscillating, time-dependent component. The introduction of this definition of the solution variables into the Navier-Stokes equations (3.4) leads to their turbulent form, which cannot be solved in practical terms because of the typically wide ranges of eddy length scales and time scales required to describe the flow [98]. Therefore, mean flow equations are derived to account for the effect of turbulence without explicitly model every single oscillation in the flow field. The latter expressions, known as Reynolds equations, introduce additional unknowns in the form of normal and tangential stresses. The resulting flow equations are the Reynolds-Averaged Navier-Stokes (RANS) equations. A thorough

discussion of their derivation can be found for example in the books of Versteeg and Malalasekera [98] and Wilcox [105]. Their index notation is the following

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ji} - \overline{\rho u'_j u'_i}) \quad (3.32)$$

where S_{ji} is the mean strain rate tensor, and $-\overline{\rho u'_j u'_i} = \tau_{ji} = \tau_{ij}$ the Reynolds stress tensor. The latter introduces additional terms that do not allow for the analytical closure of the turbulence problem [105], due to the non-linearity of the Navier-Stokes equations. In Reynolds stress models, the correlations that are unknown in the computation of the Reynolds stress tensor are expressed as a function of known flow properties in order to close the turbulence problem. The underlying assumption is that the Reynolds stress tensor is the product between the mean strain rate tensor and an eddy viscosity (Boussinesq hypothesis [106]). An additional fluid quantity, namely the turbulent viscosity μ_t , is introduced under the latter assumption. The Reynolds stress tensor for incompressible flows becomes expressed as follows

$$-\overline{\rho u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (3.33)$$

Several models have been introduced over the years to compute the turbulent viscosity. These include zero-equation models such as Prandtl's mixing length model, (zero-equation indicating that no additional transport equations are needed), one-equation models such as the Spalart-Allmaras, and two-equation models, including the k - ϵ and k - ω model. The Prandtl model relies on the concept of mixing length, in an analogy with the mean free path of molecules in a gas. It assumes that the turbulent viscosity is proportional to the mixing length scale, being valid for relatively simple cases where the latter flow-dependent quantity is known. The Spalart-Allmaras model solves one transport equation for a modified expression of the turbulent viscosity, known as kinematic eddy viscosity. The model performs well in general for external aerodynamic flows [107]; however, it is known to under-perform in the computation of strong rotating flows, for which empirical corrections are needed [108].

Two-equation models rely on the description of turbulence as composed of eddies, and on the consequent theory of energy cascade. The latter states that, in a turbulent flow, large eddies tend to break-up into progressively smaller eddies, characterised by lower kinetic energy. The energy content can be quantified as a function of the wave number $\kappa = 2\pi/\lambda$ (with λ the wave length of the eddies) through the spectral energy $E(\kappa)$ [98].

The latter is defined as the kinetic energy of the eddy per unit mass and per unit wave number at the given wave number κ . As illustrated in the plot of Figure 3.2, as the wave number κ increases, the spectral energy $E(\kappa)$ decays rapidly. The plot suggests that the energy decay at high wavelengths is linked with some form of energy dissipation in the smallest eddies. In the 1940s, Kolmogorov observed that this is connected with the viscous forces characterising them, which are the dominant forces defining their motion. As the latter forces are dissipative, Kolmogorov stated that, to maintain the energy balance in turbulent flows, the production rate of turbulent energy due to the generation of large vortices needs to be broadly in equilibrium with its rate of dissipation. Dimensional analysis led to find that the ratios between small and large velocity, length and time scales can be expressed as a function of the Reynolds number of the large eddies, thus relating microscopic flow properties with the physical behaviour at macroscopic scale. These correlations can be found for example in [98].

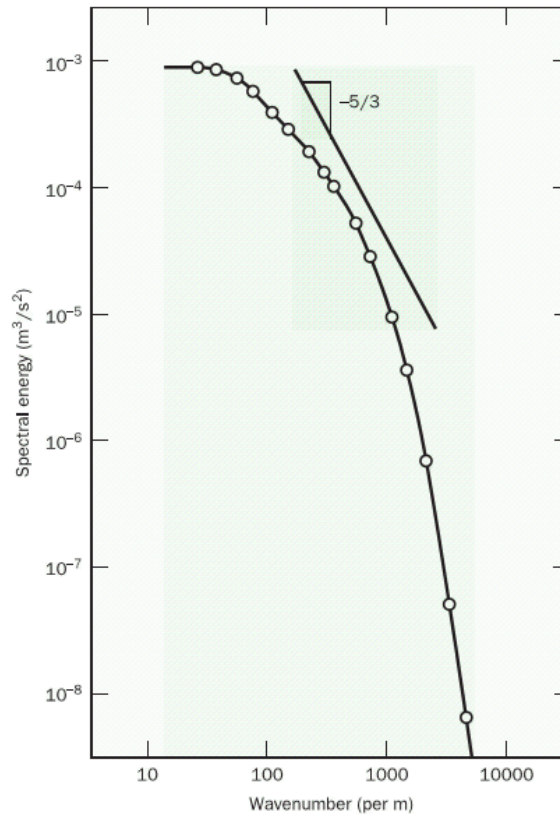


Figure 3.2 – Energy spectrum of turbulence in a grid [98]

Kolmogorov's work led to describe the turbulent viscosity as a function of the turbulence kinetic energy k , associated with the large eddies, and the turbulence dissipation rate ϵ , describing the smallest eddies. Two-equation models rely on this theory. Notably, the k - ϵ model describes the turbulent viscosity as proportional to k^2/ϵ , while the k - ω model

describes it as proportional to k/ω , with ω defined as the specific turbulence dissipation rate.

One main limitation of RANS-based models is that the behaviour of all eddies is captured by one single turbulence model. The limitation lies in the fact that the evolution of the largest eddies is highly dependent on the nature of the problem [98], which makes the development of widely applicable RANS turbulence models challenging. The Large Eddy Simulation (LES) approach addresses this problem by solving the largest eddies with a time-dependent simulation, while modelling the smallest, isotropic eddies in a way analogous to the RANS models. A cut-off length scale separates the resolved and the unresolved eddies. The latter process is known also as ‘filtering’ of the Navier-Stokes equations. The effect of unresolved eddies on the large eddies is accounted for by sub-grid-scale (SGS) stresses, which are modelled separately. Several SGS models exist, ranging from the Smagorinsky model [109], based on the Boussinesq hypothesis, to RANS-based models including transport effects [110], and more complex models such as the mixed model [111] and the dynamic SGS model [112]. The main limitations of the LES are the large computational overhead linked with the unsteady nature of the model, in particular when compared to two-equation models [98], and the strong dependency of the results on the definition of the error [113]. The latter aspect implies that the model validation is case-specific, whereas there may be limitations on data available for particular flow cases. Alternative methods to the LES include the Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations [113] and the Detached-Eddy Simulation (DES) [114].

As the simulation of two-phase flows adds a degree of complexity to the analyses performed in this thesis work, the application of turbulence models having a higher level of detail than two-equation models may lead to significant challenges in terms of computational requirements. Moreover, the lack of examples in the literature based on similar applications suggests that robust validation is yet to be achieved for the cases considered. For this reason, the modelling choices for this work have been limited to the widely-adopted two-equation models, namely the k - ϵ , k - ω , and their variants.

3.4.2 Standard k - ϵ Model

The k - ϵ model, usually defined as ‘standard’ in the form proposed by Launder and Spalding in the 1970s [115] is a widely used two-equation model which relies on the computation of turbulence kinetic energy k and its dissipation rate ϵ to calculate the turbulent viscosity μ_t of Equation (3.33). The main assumption is a fully turbulent flow.

The model solves transport equations for both k and ϵ , with the first derived from the exact equation, and the latter relying on closure approximations obtained through physical reasoning [70]. The transport equations for k and ϵ are respectively the following:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k U_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (3.34)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon U_i) \\ = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \end{aligned} \quad (3.35)$$

In Equation (3.34), the terms $G_k = 2\mu_t S_{ji} S_{ji}$ and $G_b = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}$ are respectively the generation of turbulence kinetic energy due to the mean velocity gradients and buoyancy. For the standard k - ϵ model, the turbulent Prandtl number value in the latter equation is $Pr_t = 0.85$. $\rho \epsilon$ is a sink term which represents the rate of destruction of ϵ , while S_k is a user-defined source term. The term $Y_M = 2\rho \epsilon M_t^2$ considers the effect of compressibility and can be neglected for low Mach number flows, such as in the application in this thesis.

The turbulence dissipation equation (3.35) has a form that resembles the one of the turbulence kinetic energy (3.34). Here the terms on the right hand side represent respectively the transport of ϵ by diffusion, the rate of production due to the mean velocity gradients and buoyancy, the rate of destruction of ϵ , and a user-defined source term. Both the k and ϵ equations have a structure that can be summarised as follows:

Rate of change of k or ϵ	+	Transport by convection	=	Transport by diffusion	+	Rate of production
			-	Rate of destruction	+	Source term

The turbulent (eddy) viscosity is computed from the combination of k and ϵ in the following way

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad (3.36)$$

The model constants have been tuned through data fitting over 23 turbulent flow cases, including 2D shear layers, axisymmetric jets into still and moving air, wakes, and

plumes. The original list of cases is reported in Launder et al. [116]. The model constants are provided below

$$C_{1\epsilon} = 1.44; \quad C_{2\epsilon} = 1.92; \quad C_\mu = 0.09; \quad \sigma_k = 1.00; \quad \sigma_\epsilon = 1.30 \quad (3.37)$$

3.4.3 RNG k - ϵ Model

The RNG k - ϵ model is a derivation of the standard k - ϵ model that relies on the renormalization group (RNG) technique. The latter is a statistical method used to model physical systems in which the properties are described by formulae having a similar expression at different size scales. The underlying principle is that the problem can be renormalized to solve only a sample of the system, thus reducing the size of the model. The RNG method was first implemented in turbulence modelling by Forster et al. [117] in 1977, to study the effect of a Gaussian random force on the velocity fluctuations of a flow governed by the Navier-Stokes equation. A comprehensive description of the RNG theory and its application to turbulence modelling is given in Yakhot and Orszag [118].

The RNG k - ϵ model is characterised by an extra term in the ϵ transport equation. This enhances the modelling accuracy of rapidly strained flows [98]. The model also accounts for the effects of swirl on turbulence, as well as relying on analytical formulae for some quantities which are set as constant in the standard model, as a result of the RNG theory [118]. The transport equations of k and ϵ are the following

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k U_i) = \frac{\partial}{\partial x_j} \left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (3.38)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon U_i) \\ = \frac{\partial}{\partial x_j} \left[\alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} - R_\epsilon \\ + S_\epsilon \end{aligned} \quad (3.39)$$

where the terms G_k and G_b are computed as in the standard model (Section 3.4.2), and Y_M can be neglected for incompressible flows.

Equations (3.38) and (3.39) differ from the standard k - ϵ model for the use of $\alpha_{k,\epsilon} \mu_{eff}$ instead of $\mu + \frac{\mu_t}{\sigma_{k,\epsilon}}$ in the term representing the transport by diffusion of k and ϵ , i.e. the first term on the right hand side of the two equations. α_k and α_ϵ are the inverse effective

Prandtl numbers for k and ϵ , and are computed with the following equation obtained analytically from the RNG theory

$$\left| \frac{\alpha - 1.3929}{\alpha_0 - 1.3929} \right|^{0.6321} \left| \frac{\alpha + 2.3929}{\alpha_0 + 2.3929} \right|^{0.3679} = \frac{\mu_{mol}}{\mu_{eff}} \quad (3.40)$$

with $\alpha_0 = 1.0$. The effective turbulent viscosity μ_{eff} is computed by integrating the following differential equation resulting from the RNG theory

$$d \left(\frac{\rho^2 k}{\sqrt{\epsilon \mu}} \right) = 1.72 \frac{\hat{v}}{\sqrt{\hat{v}^3 - 1 + C_v}} d\hat{v} \quad (3.41)$$

where

$$\hat{v} = \frac{\mu_{eff}}{\mu} \quad (3.42)$$

and $C_v \approx 100$. For high Reynolds numbers, Equation (3.41) becomes the same expression of μ_t of the standard k - ϵ model (Equation (3.36)), although the constant C_μ has a value of 0.0845 instead of the empirically-determined 0.09.

In Equation (3.39) the additional term R_ϵ is introduced to account for rapidly strained flows. It is defined as

$$R_\epsilon = \frac{C_\mu \rho \eta^3 \left(1 - \frac{\eta}{\eta_0} \right) \epsilon^2}{1 + \beta \eta^3} \frac{1}{k} \quad (3.43)$$

with $\eta \equiv \frac{Sk}{\epsilon}$, $\eta_0 = 4.38$, $\beta = 0.012$. In the regions characterised by a high strain rate, i.e. $\eta > \eta_0$, the term R_ϵ decreases the destruction of ϵ , leading to a reduction of effective viscosity and thus improving the performance of the k - ϵ model for rapid strain and high curvatures in the flow [70].

A modification for flows characterised by high swirl is also available in the model. Further details on its implementation are available in [70]. The model constants are the following:

$$C_{1\epsilon} = 1.42; \quad C_{2\epsilon} = 1.68 \quad (3.44)$$

3.4.4 k - ω Model

The k - ω is a two-equation turbulence model based on the approach proposed by Wilcox [105]. It accounts for the effects of low Reynolds numbers, compressibility, and shear flow spreading. In the formulation reported in this section the terms related with compressibility are not included, as the latter effect is negligible for the flow conditions

analysed in this work. The basic structure of the model is similar to the k - ϵ (Section 3.4.2), as two transport equations are solved to compute the turbulent viscosity needed to close the RANS equations (3.32). The k - ω model differentiates itself from the k - ϵ in that the specific dissipation rate (ω) is used instead of the dissipation rate (ϵ). The transport equations are the following

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k U_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (3.45)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega U_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega \quad (3.46)$$

where G, Y, S are respectively the rate of production, the rate of destruction, and the user-defined source term for the transport equations. In (3.46), the dissipation of ω is given by

$$Y_\omega = \rho \beta f_\beta \omega^2 \quad (3.47)$$

where

$$f_\beta = \frac{1 + 70\chi_\omega}{1 + 80\chi_\omega} \quad (3.48)$$

$$\chi_\omega = \left| \frac{\Omega_{ij}\Omega_{jk}S_{ki}}{(\beta_\infty^*\omega)^3} \right| \quad (3.49)$$

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \quad (3.50)$$

For the description of the terms G_k, Y_k, G_ω in Equations (3.45)-(3.46) it is recommended to refer to the available literature, for example [105].

The turbulent viscosity is computed as

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (3.51)$$

thus in a similar form to the expression of the k - ϵ model (3.36). In (3.51), the coefficient α^* damps the turbulence viscosity at low Reynolds numbers, and is equal to 1 as $Re_t = \frac{\rho k}{\mu \omega} \rightarrow \infty$.

The model constants are summarised below

$$\alpha_\infty^* = 1; \alpha_\infty = 0.52; \alpha_0 = \frac{1}{9}; \beta_\infty^* = 0.09; \beta_i = 0.072; R_\beta = 8; R_k = 6; R_\omega = 2.95; \zeta^* = 1.5; \sigma_k = 2.0; \sigma_\omega = 2.0 \quad (3.52)$$

3.4.5 SST k - ω Model

The Shear Stress Transport (SST) modification of the k - ω model was introduced by Menter [119] to improve the prediction accuracy of flows where separation occurs and significant adverse pressure gradients are present [120]. In principle, the SST model relies on blending functions to switch between the near-wall formulation of the k - ω model and the freestream formulation of the k - ϵ model. Following this approach, the shortcomings of the k - ϵ model in the solution of the near-wall layers and those of the standard k - ω model in the solution of freestream regions are overcome [120]. As the SST model is based on a zonal formulation, the computation of the wall distance is required, thus adding complexity to the model compared to the standard k - ϵ and k - ω .

The SST k - ω transport equations have a similar form to those of the standard k - ω model

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k U_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \quad (3.53)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega U_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega \quad (3.54)$$

Comparing the form of Equations (3.53)-(3.54) with that of (3.45)-(3.46), the only additional term in the SST k - ω transport equations is the cross-diffusion D_ω , which is defined to blend the k - ϵ model into the k - ω formulation. It is expressed as

$$D_\omega = 2(1 - F_1)\rho \frac{1}{\omega \sigma_{\omega,2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (3.55)$$

where

$$F_1 = \tanh(\Phi_1^4) \quad (3.56)$$

$$\Phi_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right), \frac{4\rho k}{\sigma_{\omega,2} D_\omega^+ y^2} \right] \quad (3.57)$$

$$D_\omega^+ = \max \left[2\rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right] \quad (3.58)$$

where y is the distance to the next surface and D_ω^+ the positive part of D_ω . The dissipation term Y_ω is defined as in Equation (3.47), with $f_\beta = 1$, thus

$$Y_\omega = \rho \beta \omega^2 \quad (3.59)$$

although instead of being constant, β is defined here as:

$$\beta = F_1\beta_{i,1} + (1 - F_1)\beta_{i,2} \quad (3.60)$$

Further modifications compared to the standard k - ω model include the expressions of the turbulent Prandtl numbers σ_k and σ_ω , which are no longer constant, and the expression of G_ω . For the description of these parameters, it is recommended to refer to [119].

The turbulent viscosity, which accounts for the transport of the turbulence shear stress, is defined as

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1\omega}\right]} \quad (3.61)$$

where S is the strain rate magnitude. F_2 is given by

$$F_2 = \tanh(\Phi_2^2) \quad (3.62)$$

$$\Phi_2 = \max\left(2 \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega}\right) \quad (3.63)$$

The model constants are the following:

$$\begin{aligned} \sigma_{k,1} = 1.176; \quad \sigma_{\omega,1} = 2.0; \quad \sigma_{k,2} = 1.0; \quad \sigma_{\omega,2} = 1.168; \quad \alpha_1 \\ = 0.31; \quad \beta_{i,1} = 0.075; \quad \beta_{i,2} = 0.0828 \end{aligned} \quad (3.64)$$

All the remaining constants required have the same values as in the standard k - ω model.

The major advantage of the SST k - ω model for the problem analysed in this thesis is its applicability to a wide range of wall y^+ values, which gives a relatively low sensitivity of the results to the near-wall mesh resolution [120]. This is a case particularly relevant where fast moving parts are in proximity of stationary walls, such as in bearing chambers, and where geometries are particularly complex to maintain a uniform, limited range of wall y^+ values in the whole domain (see Section 3.4.6). Our preliminary tests performed on the KIT bearing chamber, reported in Section 4.2.5, confirmed the validity of this model in computing correctly the wall-fluid momentum transfer for different wall y^+ values. For the reasons above, the SST k - ω model has been adopted for the majority of the simulations presented in this thesis.

3.4.6 Near-wall Modelling: Momentum Transfer

The presence of solid walls in contact with the fluid has a significant effect on turbulent flows. As on the wall surface the fluid is static, high velocity gradients are generally present in the region in proximity of the wall. This near-wall region is known as

boundary layer, and can be divided into an inner region, composed of three sublayers, and an outer region, as illustrated in Figure 3.3. In the first sublayer, i.e. the closest to the wall, the flow tends to behave as laminar due to the high viscosity linked with the high tangential velocity gradients. This region is characterised by $\mu_t \ll \mu$, and is normally referred as viscous sublayer. Moving further from the wall, a buffer zone characterised by a transition from laminar to turbulent flow is found ($\mu_t \approx \mu$), followed by a region where turbulence has a significant effect on the flow behaviour ($\mu_t \gg \mu$). This part of the boundary layer is divided into the log law region and the outer region.

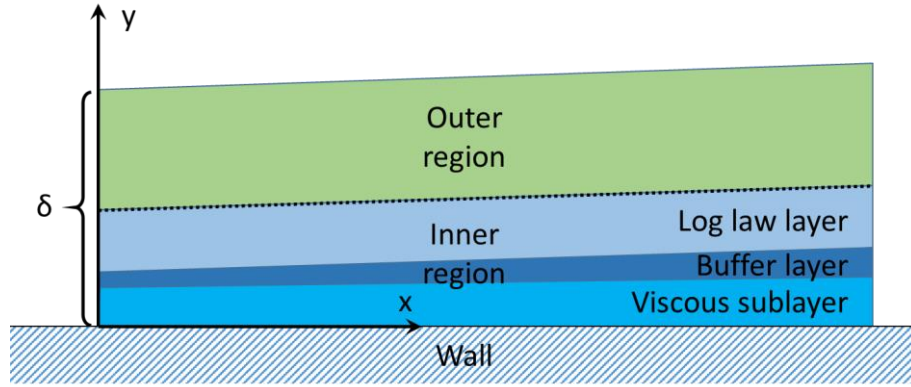


Figure 3.3 – Boundary layer structure

To better evaluate the turbulent characteristics of the flow depending on the wall distance, the non-dimensional quantities y^+ and u^+ are typically used,

$$y^+ = \frac{\rho U_\tau y}{\mu} \quad (3.65)$$

where U_τ is the shear velocity, proportional to the wall shear stress τ_w ,

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (3.66)$$

and y the distance from the wall. The non-dimensional velocity u^+ is defined as

$$u^+ = \frac{u}{U_\tau} \quad (3.67)$$

Considering its formulation and meaning, the y^+ number can be regarded as a flow Reynolds number based on the distance from the wall. Low y^+ values ($y^+ < 30$) characterise the viscous sublayer, in which the mean flow velocity depends only on the variables defining y^+ in Equation (3.65) [98]. In the region $30 < y^+ < 500$ instead the mean flow velocity is proportional to the natural logarithm of the wall distance as follows:

$$u^+ = \frac{1}{\kappa} \ln y^+ + C \quad (3.68)$$

In (3.68), $\kappa = 0.41$ is the Karman constant. C ranges between 5.5 and 8.5; it is derived from data fitting and it is function of the local roughness, with $C \approx 5.5$ for a hydraulically smooth surface. Equation (3.68) is known also as log law. Figure 3.4 reports the velocity distribution of a turbulent boundary layer next to a wall, showing the range in which the log law is valid.

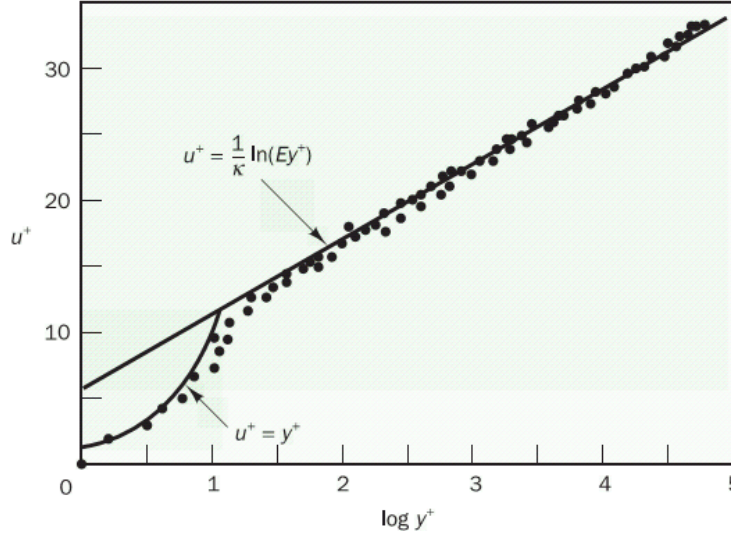


Figure 3.4 – Velocity distribution in proximity of a solid wall [98]

In the outer region of the boundary layer ($y/\delta > 0.2$) [98], the log law is no longer valid. In this region the flow is dominated by the inertia of the core flow, and the law of the wake is used to describe its mean velocity

$$\frac{U_{max} - U}{U_\tau} = \frac{1}{\kappa} \ln \left(\frac{y}{\delta} \right) + A \quad (3.69)$$

where A is a constant [98].

In CFD, two approaches are generally adopted to model the inner region of the boundary layer. The first method, used for example in the standard $k-\omega$ model, is to resolve fully the boundary layer with a fine computational grid, extended through the viscous sublayer. This technique implies that a certain number of cells, of the order of 10, needs to be placed in the region $y^+ < 30$. For high Reynolds number flows, however, the boundary layer can be extremely thin compared to the geometry to be modelled, meaning that a large number of high aspect ratio cells may be required to resolve the viscous sublayer. Moreover, the cell size needs to grow away from the wall at a finite rate, typically

‘geometrically’, in order to avoid large discontinuities. Since this can lead to very large mesh requirements and convergence issues arising from high cell aspect ratios, a second approach based on wall functions is commonly used in CFD. With this second method, implemented in the k - ϵ model, the flow velocity profile in the inner boundary layer is computed based on the correlations seen in Equations (3.65)–(3.68). The first cell centroid is placed in the log law region, at $y^+ \approx 30$, and the wall shear velocity U_τ is computed from the velocity value u in the centroid from

$$\frac{u}{U_\tau} = \frac{1}{\kappa} \ln \left(\frac{y U_\tau}{\nu} \right) + C \quad (3.70)$$

The production and dissipation rates of the turbulence kinetic energy in this cell are then estimated. In the region above $y^+ \approx 30$, the log law (3.68) is used to compute the mean flow velocity until the end of the inner boundary layer, from which the equations of the standard or RNG k - ϵ model (Sections 3.4.2 and 3.4.3) are used.

The wall function approach described above is known as standard wall function. In some flow cases, mesh constraints can make challenging to meet its wall y^+ requirements in the whole domain. For this reason, some models such as scalable wall functions and the enhanced wall treatment incorporate adjustments to meet the required conditions. Scalable wall functions address cases in which the centroid of the near-wall cell falls within the viscous sublayer, i.e. its y^+ value is too low. The correction introduces a limiter as follows,

$$\tilde{y}^+ = \max(y^+, y_{limit}^+) \quad (3.71)$$

with typically $y_{limit}^+ = 11.225$ [70]. Examples of cases requiring this approach include the analysis of axial compressor fouling, where arbitrarily fine meshes near the blade may lead to very low y^+ values [121]. The enhanced wall treatment relies instead on a two-layer formulation with a blending function to resolve explicitly both the viscous sublayer (where required) and the turbulent boundary layer. The two layers are identified following a wall distance-based, turbulent Reynolds number criterion. This formulation is useful in flow cases where a wide range of y^+ values characterises the model, such as cavities next to the main gas path in turbomachinery components [122].

Although the use of wall functions is typical of k - ϵ models, near-wall corrections are available also in k - ω models. For example, ANSYS Fluent relies on an additional function that mimics the law of the wall to preserve the accuracy of the results where excessive wall y^+ values are found. The description of this formulation is available in [70].

When defining the numerical model, it should be kept in mind that the near-wall mesh sizing might depend on additional requirements such as the need to capture thin liquid layers and the thermal boundary layer. In bearing chamber applications, these requirements lead normally to y^+ values lower than 10 on a large portion of the walls of the domain. Therefore, in the simulations presented in this work the viscous sublayer is generally explicitly resolved.

3.4.7 Near-wall Modelling: Heat Transfer

In ANSYS Fluent, the modelling of heat transfer in turbulent boundary layers relies on the so-called Reynolds analogy. The latter is based on the assumption that, in turbulent boundary layers, the energy transfer takes place mainly through the interchange of fluid particles [123], as illustrated in Figure 3.5.

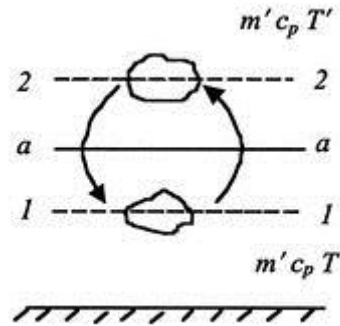


Figure 3.5 – Momentum and heat exchange in turbulent boundary layer [124]

With reference to Figure 3.5, in a steady flow there is a correspondence in the mass transfer between plane 1 – 1 and 2 – 2, meaning that for a generic fluid particle having mass m' going from plane 1 – 1 to 2 – 2 there is an equivalent particle with mass m' moving downwards to plane 1 – 1. The two particles have temperature T and T' respectively. Due to their exchange, the turbulent heat flux q_t through the plane $a - a$ will be

$$q_t = m' c_p (T' - T) \quad (3.72)$$

where c_p is the specific heat at constant pressure of the fluid particle.

In their motion, the two fluid particles have tangential components of their momentum $m' U_\perp$ and $m' U'_\perp$, where $U_\perp$ and $U'_\perp$ are respectively their tangential velocities per unit time. Relative to plane $a - a$, the momentum exchange per unit area and time can be defined as a shear stress,

$$\tau_t = m'(U'_\perp - U_\perp) \quad (3.73)$$

Combining (3.72) and (3.73),

$$q_t = \tau_t c_p \left(\frac{T' - T}{U'_\perp - U_\perp} \right) \quad (3.74)$$

which in differential form, as the location of planes 1 – 1 and 2 – 2 is arbitrary, becomes

$$q_t = -\tau_t c_p \left(\frac{dT}{dU_\perp} \right) \quad (3.75)$$

In the laminar sublayer of the boundary layer, heat is transferred in the direction orthogonal to the wall through conduction. Therefore, the law describing the heat flux is

$$q_{lam} = -k \frac{dT}{dy} \quad (3.76)$$

where k is the thermal conductivity of the fluid and y the direction normal to the wall.

Dividing (3.76) by $\tau = \mu \frac{dU_\perp}{dy}$, the following expression is obtained,

$$q_{lam} = -\tau \frac{k}{\mu} \frac{dT}{dU_\perp} \quad (3.77)$$

Comparing (3.75) and (3.77), and assuming that $\tau_t = \tau$, it is observed that if $c_p = k/\mu$, i.e. the Prandtl number $Pr = \frac{\mu c_p}{k} = 1$, the turbulent and laminar heat fluxes can be expressed in the same way, thus

$$q = -\tau c_p \left(\frac{dT}{dU_\perp} \right) \quad (3.78)$$

hence,

$$dT = -\frac{q}{\tau c_p} dU_\perp \quad (3.79)$$

Equation (3.79) can be then integrated from the wall to the free stream. On the hypothesis that q/τ is constant, this leads to

$$q_w \approx \frac{\tau_w c_p (T_w - T_\infty)}{U_\infty} \quad (3.80)$$

which is known as the Reynolds analogy between heat transfer and shear stress.

As stated above, the Reynolds analogy is valid on the hypothesis that the Prandtl number of the fluid is unity, which means that the thermal and fluid boundary layers have the same thickness. This is valid in general for gases [124], but not in the case of liquids.

For example, the oil used in the simulations in this thesis has $Pr \approx 80$. Modifications to the Reynolds analogy have been proposed in the literature to account for different fluid Prandtl numbers. One example is the Chilton-Colburn analogy [123], introduced in 1934. The implementation in ANSYS Fluent [70] takes into account the ratio between fluid and thermal boundary layer thicknesses, which is

$$\frac{\delta}{\delta_T} \approx \sqrt{Pr} \quad (3.81)$$

and implements the following thermal law of the wall

$$T^* \equiv \frac{(T_w - T_P)\rho c_p u^*}{q_w} = T_c^* + \frac{D}{q_w} \quad (3.82)$$

where T^* is the non-dimensional fluid temperature, the subscript P indicates the centroid of the wall-adjacent cell, and $u^* = C_\mu^{0.25} k_P^{0.5}$. When the viscous heating effects are small compared to the wall heat flux, the term D/q_w in (3.82) is neglected, thus $T^* \approx T_c^*$. The non-dimensional convective-conductive temperature T_c^* is computed as follows

$$T_c^* = \begin{cases} Pr y^* & \text{if } y^* < y_T^* \\ Pr_t \left[\frac{1}{\kappa} \ln(E y^*) + P \right] & \text{if } y^* > y_T^* \end{cases} \quad (3.83)$$

where y_T^* is the non-dimensional thermal sublayer thickness, computed as the value of y^* at which the conductive (linear) and convective (logarithmic) laws in (3.83) intersect. In Equation (3.83), $E = 9.793$ is an empirical constant, while P is computed as

$$P = 9.24 \left[\left(\frac{Pr}{Pr_t} \right)^{0.75} - 1 \right] [1 + 0.28 e^{-0.007 Pr / Pr_t}] \quad (3.84)$$

The computational procedure to apply the thermal law of the wall is therefore the following:

1. Computation of molecular Prandtl number (Pr) from fluid physical properties;
2. Computation of thermal sublayer thickness y_T^* ;
3. Application of either linear or logarithmic profile of Equation (3.83) to compute T_c^* ;
4. Computation of either the wall heat flux q_w or the wall temperature T_w through Equation (3.82), depending on the thermal boundary conditions.

3.5 Turbulence Damping

3.5.1 Overview

Since the Volume of Fluid formulation relies on shared quantities such as velocity and turbulence in each cell of the domain, the presence of an interface between two phases does not have any physical effect in the cell where the interface lies. The flow quantities are simply averaged as shown in Section 3.3.1, thus information about the momentum transfer between two phases, according to the position and orientation of their interface within a cell, will be lost. This is generally not a problem for the majority of flows found in industrial applications, but becomes relevant when a liquid phase is driven by the shear forces of a gas in contact with its surface. In this case, a non-physical momentum transfer takes place between the gas and the liquid when using differential turbulent viscosity models such as the k - ϵ or k - ω [80].

Egorov [80] at ANSYS CFX was the first to introduce a wall-like turbulence damping at the gas-liquid interface of shear-driven flows, for a two-fluid method. His damping term overcomes the empirical approach of previous models [75, 79] to extend the applicability to a wider range of flow conditions. These include cases in which the liquid layer thickness and surface wave properties are not known in advance [80]. Having proven the enhancement in accuracy of the predicted velocity profiles in both the gas and liquid phase [1], this approach was taken up at Nottingham by Tkaczyk et al. [42] for implementation in the VOF method. Since then, it has become available in ANSYS Fluent for their k - ω model.

As discussed in Section 2.3.4, alternative approaches to the one proposed by Egorov [80] include the Simplified Interface Treatment (SIT) and Mathematically Closed Interface Treatment (MCIT) presented by Hashmi in his PhD thesis [40] at the KIT. These two methods were developed specifically for bearing chamber flows, although for the MCIT there are no examples of application to bearing chambers available in the literature. Therefore, one of the objectives of this thesis is to verify the accuracy and benefits of the latter method for bearing chamber modelling (see Section 4.4.4).

3.5.2 Egorov's Turbulence Damping

The turbulence damping approach followed by Egorov [80] modifies the ω equation of the k - ω turbulence model to introduce a solid wall-like damping at the gas-liquid

interface. The modification was developed and tested for a stratified, smooth, shear-driven liquid flow in a rectangular channel.

The assumptions of Egorov's approach [80] are a smooth free surface and no account for the anisotropy between the normal and tangential Reynolds stresses in the liquid phase. Hence, the model introduces a symmetric damping between the two phases. In order to obtain a wall-like behaviour at the interface, the wall Dirichlet boundary condition of the k - ω model was identified as

$$\omega_w = B \cdot \frac{6\mu}{\beta\rho\Delta n^2} \quad (3.85)$$

where B is a scaling coefficient which should be at least 10, μ the dynamic viscosity, β the closure coefficient of the destruction term, and Δn the near-wall cell height. To reproduce the effect of Equation (3.85) on a free surface, Egorov [80] added the following source term on the right hand side of the ω equation (3.46), for both the gas and liquid phases,

$$A \cdot \Delta y \beta \rho_i \left(B \cdot \frac{6\mu_i}{\beta \rho_i \Delta n^2} \right)^2 \quad (3.86)$$

where $A = |\nabla \alpha_g|$ is the interface area density and the subscript i represents the i -th phase. The factor A ensures that the source term is activated only at the free surface to cancel the destruction term $(-\alpha_i \beta \rho_i \omega_i^2)$ (3.47) of the ω equation (3.46). This increases the value of ω_i , hence the turbulence damping.

The effect of turbulence damping is clearly visible on the velocity profiles of the stratified gas-liquid channel flow tested by Egorov [80], reported in Figure 3.6. On the left hand side, the turbulence damping function is not active, thus an excessive momentum transfer takes place from the gas to the liquid phase. The right hand side figure shows the results obtained with turbulence damping. In this case, a steep velocity gradient at the interface between the phases is clearly distinguishable, consistently with the wall-like behaviour imposed on the liquid surface.

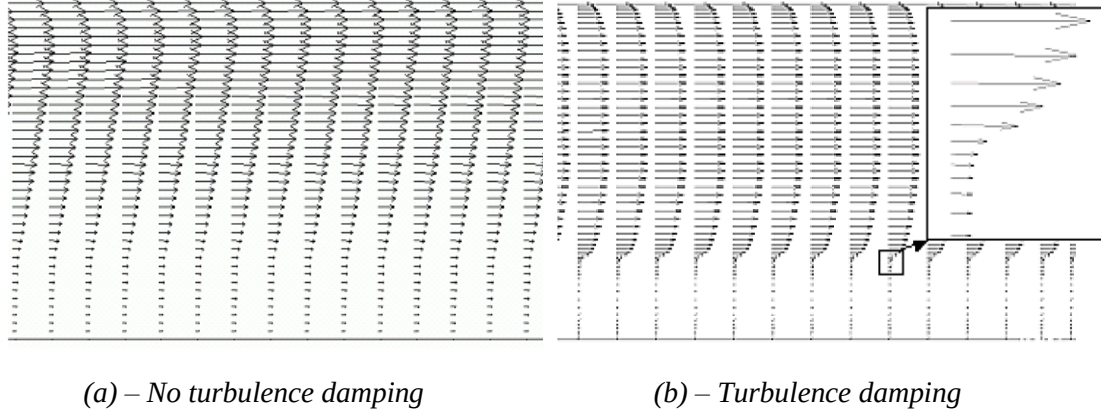


Figure 3.6 – Effect of interface turbulence damping on the velocity profiles of a stratified gas-liquid flow in a rectangular channel [80]

The approach of Egorov [80] requires a grid capable of resolving the viscous sub-layer at the free surface. Although this is ensured ideally by $y^+ = 1$, the numerical tests performed by Egorov [80] relied on a coarser mesh at the interface ($y^+ = 40$). The outcome was a recommended value of 100 for the scaling coefficient B , due to the relatively coarse grid [80]. The mesh-dependence of B and the need to tune this parameter to match the experimental data has been identified in this thesis as one of the major limitations of this turbulence damping approach.

3.5.3 Hashmi's Mathematically Closed Interface Treatment

Hashmi's work [39] attempts to overcome the need to choose a mesh-dependent scaling factor, required in methods such as Egorov's turbulence damping approach [80]. While the latter modifies the ω equation of the k - ω model to obtain the Dirichlet boundary condition of a solid wall at the gas-liquid interface, Hashmi's Mathematically Closed Interface Treatment (MCIT) starts from the assumption that the turbulent viscosity near the interface should tend to zero. This is based on the hypothesis that a laminar sub-layer exists near the free surface. In this layer, viscous stresses dominate the flow and turbulent stresses are negligible [39].

Hashmi's MCIT derivation starts from the turbulent viscosity formulation of the k - ϵ model, i.e. Equation (3.36). The equation is then expressed as a function of ϵ and differentiated with respect to time, to obtain,

$$\frac{\partial \epsilon}{\partial t} = 2\rho k \frac{C_\mu}{\mu_t} \frac{\partial k}{\partial t} + k^2 \frac{C_\mu}{\mu_t} \frac{\partial \rho}{\partial t} = 2k \frac{C_\mu}{\mu_t} \frac{\partial(\rho k)}{\partial t} \quad (3.87)$$

on the hypothesis that $\frac{\partial \rho}{\partial t} = 0$, as the term is updated in the solver at each time step [39].

Considering now the turbulent kinetic energy equation of the k - ϵ model as reported in Versteeg and Malalasekera [98] for example, the convection and diffusion transport terms at the interface are zero as the turbulent kinetic energy production in this region is assumed to be zero. The rate of destruction term ($-\rho\epsilon$) is also neglected, as it is already accommodated in Equation (3.36) [39]. Therefore, the k equation at the free surface becomes

$$\frac{\partial(\rho k)}{\partial t} = 2\mu_t S_{ji} \cdot S_{ji} \quad (3.88)$$

where E_{ij} is the mean rate of deformation tensor. Equation (3.88) can now be replaced into (3.87), leading to,

$$\frac{\partial\epsilon}{\partial t} = 4kC_\mu S_{ji} \cdot S_{ji} \quad (3.89)$$

Multiplying both sides of (3.89) by ρ , the following expression is obtained

$$\frac{\partial\rho\epsilon}{\partial t} = 4k\rho C_\mu S_{ji} \cdot S_{ji} \quad (3.90)$$

which can be added to the ϵ equation of the k - ϵ model. This term introduces the required turbulence dissipation to obtain the correct near-wall turbulent viscosity value. In order to have this term only at the interface, an activation function is introduced,

$$\Lambda = \frac{A - 1}{A + 1} \cdot \alpha_g \quad (3.91)$$

where A is the interface area density and α_g the gas phase volume fraction. The final source term to be added in the ϵ equation becomes therefore:

$$Source = 4\Lambda k\rho C_\mu S_{ji} \cdot S_{ji} \quad (3.92)$$

In the work of Hashmi [39], a source term for the k - ω model is also available. For comparison purposes with the test cases proposed in [39], however, the k - ϵ source term of Equation (3.92) has been implemented in this work.

Hashmi [39] proposed in his work also a simpler turbulence damping approach called Simplified Interface Treatment (SIT). With the latter approach, the turbulent viscosity is directly modified as follows:

$$\mu_t = \begin{cases} \approx 0 & \text{if } 0 < \alpha_g < 1 \\ \rho C_\mu \frac{k^2}{\epsilon} \text{ or } \rho \frac{k}{\omega} & \text{if } \alpha_g = 0 \text{ or } \alpha_g = 1 \end{cases} \quad (3.93)$$

With this approach, the flow is set as laminar in the region where the interface is located, thus reproducing the characteristics of the viscous sublayer. Despite the promising results presented by Krug [92] for a coarse mesh, in the tests performed in this thesis work this approach generated numerical instabilities. The instabilities were linked to the introduction of a discontinuity in the turbulent viscosity field at the interface with Equation (3.93), combined with very high aspect ratio cells that did not favour convergence in that region. Attempts to smooth-out the field with additional functions were performed, but did not lead to satisfactory results. The outcome led to the conclusion that the SIT approach requires additional work for its stable implementation.

3.6 Discretisation and Solution Method

3.6.1 Introduction

This section provides an overview of the discretisation and solution algorithms adopted in the simulations presented in this thesis, in order to provide a reference to the results and methods reported in Chapters 4 and 5. Since a full review of the numerical methods and solvers available in CFD would go beyond the purpose of this thesis, only the relevant methods implemented in this work are summarised in this section. For further reference, a comprehensive discussion of the numerical methods available for CFD codes and of their implementation can be found in the book of Versteeg and Malalasekera [98].

3.6.2 The Finite Volume Method

Transport Equation

As the conservation equations expressed in Section 3.2 and 3.3 are non-linear, second order partial differential equations, their numerical solution is based on a discretisation procedure. Their formulation is defined for a generic Eulerian control volume, thus the domain is divided in multiple volumes in which the conservation equations are solved. This method is known as Finite Volume Method (FVM), and is adopted for all our simulations.

In the solution of a fluid dynamics problem in a control volume, the conservation equations of a generic quantity $\Phi = \Phi(x, y, z, t)$ are integrated over the volume V as follows

$$\begin{aligned}
\int_V \frac{\partial \rho \Phi}{\partial t} dV + \int_V \nabla \cdot (\rho \Phi \mathbf{U}) dV \\
= \int_V \nabla \cdot (\Gamma_\Phi \nabla \Phi) dV + \int_V S_\Phi dV
\end{aligned} \tag{3.94}$$

where Γ_Φ is a diffusion coefficient, and S_Φ a local source term. Using the Gauss's divergence theorem, Equation (3.94) becomes

$$\int_V \frac{\partial \rho \Phi}{\partial t} dV + \oint_S \rho \Phi \mathbf{U} \cdot d\mathbf{A} = \oint_S \Gamma_\Phi \nabla \Phi \cdot d\mathbf{A} + \int_V S_\Phi dV \tag{3.95}$$

where S is the surface of the control volume, and $\mathbf{A}$ the vector area.

Discretising the problem into a given control volume having N faces, the transport equation (3.95) becomes

$$\frac{\partial \rho \Phi}{\partial t} V + \sum_{f=1}^N \rho_f \Phi_f \mathbf{U}_f \cdot \mathbf{A}_f = \sum_{f=1}^N \Gamma_\Phi \nabla \Phi_f \cdot \mathbf{A}_f + S_\Phi V \tag{3.96}$$

where Φ_f is the value of Φ flowing convectively in the face f , $\nabla \Phi_f$ the gradient of Φ at the face f , and $\mathbf{A}_f$ the vector area of f . The quantity $\rho_f \mathbf{U}_f \cdot \mathbf{A}_f$ corresponds to the mass flow through f .

Spatial Discretisation

In ANSYS Fluent, the information about each control volume (or cell) is stored in its centroid, i.e. C_0 and C_1 as illustrated in the example of Figure 3.7 for two arbitrary 2D cells. In addition to the value of Φ stored for each volume, the face values Φ_f are required to compute the convective terms in Equation (3.96). These values are obtained using the quantities of the cell upstream (known as upwind), according to the chosen numerical scheme [70].

In this work, the second order upwind scheme has been selected for the spatial discretisation of the momentum, k , ω or ϵ , and energy equations. This scheme computes the face value Φ_f from the cell upstream using its centroid value Φ_U and its gradient as follows:

$$\Phi_f = \Phi_U + \nabla \Phi_U \cdot \mathbf{r} \tag{3.97}$$

where $\mathbf{r}$ is the vector of displacement from the upstream cell centroid to the centroid of the face f , as shown in Figure 3.7.

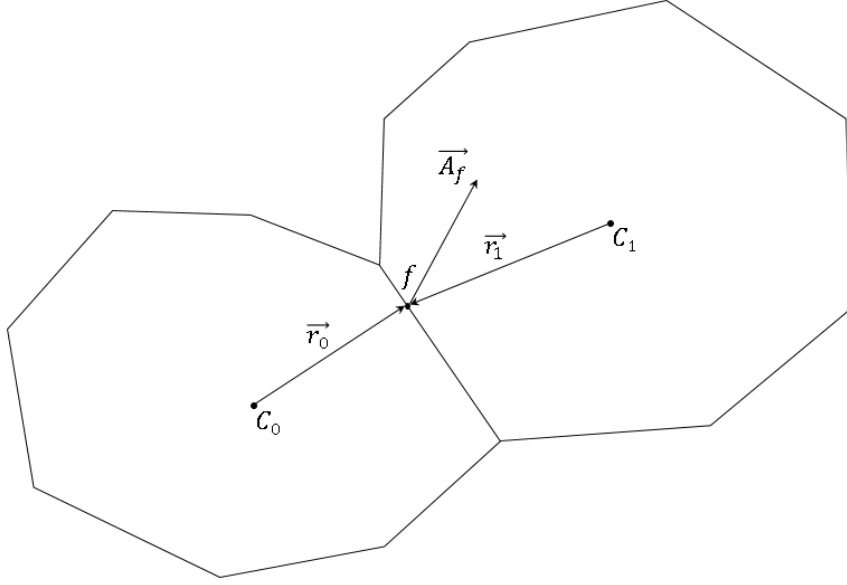


Figure 3.7 – Control volumes for discretisation of scalar transport equation

Time discretisation

The time-derivative term in Equation (3.96), i.e. $\frac{\partial \rho \Phi}{\partial t} V$, requires discretisation over time. In the approach implemented in ANSYS Fluent, the integration of this term over time is made through a backward-differencing scheme, which can be either of the first order or higher. Considering the generic expression for the scalar quantity Φ ,

$$\frac{\partial \Phi}{\partial t} = F(\Phi) \quad (3.98)$$

where F includes any spatial discretisation, the first order time discretisation is performed as follows

$$\frac{\Phi^{n+1} - \Phi^n}{\Delta t} = F(\Phi) \quad (3.99)$$

while the second order is

$$\frac{3\Phi^{n+1} - 4\Phi^n + \Phi^{n-1}}{2\Delta t} = F(\Phi) \quad (3.100)$$

where $n - 1$, n , and $n + 1$ represent respectively the values at the previous ($t - \Delta t$), current (t), and next ($t + \Delta t$) time step.

When performing an implicit time integration, (3.99) is expressed as

$$\frac{\Phi^{n+1} - \Phi^n}{\Delta t} = F(\Phi^{n+1}) \quad (3.101)$$

thus the function F is evaluated at the successive time step. The explicit formulation evaluates F at the current time step n instead

$$\frac{\Phi^{n+1} - \Phi^n}{\Delta t} = F(\Phi^n) \quad (3.102)$$

All the simulations presented in this work adopt either the first order implicit or bounded second order implicit time discretisation. The latter uses the value of F computed at the time level $n + \frac{1}{2}$ to evaluate the time evolution of Φ as follows

$$\frac{\Phi^{n+\frac{1}{2}} - \Phi^{n-\frac{1}{2}}}{\Delta t} = F\left(\Phi^{n+\frac{1}{2}}\right) \quad (3.103)$$

where

$$\Phi^{n+\frac{1}{2}} = \Phi^n + \frac{1}{2}\beta^{n+\frac{1}{2}}(\Phi^n - \Phi^{n-1}) \quad (3.104)$$

$$\Phi^{n-\frac{1}{2}} = \Phi^{n-1} + \frac{1}{2}\beta^{n-\frac{1}{2}}(\Phi^{n-1} - \Phi^{n-2}) \quad (3.105)$$

In the latter two equations, the factors β are bounding factors for each variable at the time level $n + \frac{1}{2}$ and $n - \frac{1}{2}$. The bounded variables include the volume fraction α , the turbulence kinetic energy k , and the turbulence dissipation rate ϵ or specific dissipation rate ω .

Time step Selection

When using explicit time integration, the stability of the solution is based on satisfying the Courant-Friedrichs-Lewy (CFL) law. The latter states that for a given computational cell having a characteristic length Δx along which the flow moves with average velocity U , the following condition needs to be satisfied,

$$CFL = \frac{U \Delta t}{\Delta x} \leq 1 \quad (3.106)$$

thus the time step size Δt should be chosen accordingly. When using the implicit time integration instead, the solution is stable independently of the choice of the time step. Nonetheless, the CFL criterion can be used as a guidance to select the initial time step size, in order to avoid excessively large time steps that may prevent convergence despite relying on the implicit time integration.

3.6.3 Solution Algorithms

The numerical methods implemented in ANSYS Fluent to compute iteratively the solution are divided into pressure- or density-based. The first method was originally developed for incompressible flows, while the second is normally used for high-speed flows which include compressibility effects [70]. In both methods the velocity is computed from the momentum equation (3.4), while the calculation of density and pressure is different between the two approaches. In the density-based, the pressure is computed from the equation of state and the density from the continuity equation. In the pressure-based method instead, since density is assumed constant, the pressure is obtained from a pressure correction equation derived from the continuity and momentum equations. The solution procedure for either method is the following:

1. Discretisation of the domain into control volumes;
2. Integration of the governing equations of each control volume to generate algebraic equations for the unknown variables, (e.g. pressure, velocity...), as discussed in Section 3.6.2;
3. Construction of a system of linear equations for the unknown variables, of the form $a_p \Phi = \sum_{nb} a_{nb} \Phi_{nb} + b$, where nb indicates the neighbouring cells, and a_p and a_{nb} are the linearised coefficients for the quantities Φ and Φ_{nb} ;
4. Solution and update of the variables.

All the simulations presented in this work rely on the pressure-based solver, as the compressibility effects at the given flow velocities are negligible. Nonetheless, the fluid densities are updated between each simulation according to the changes of operating condition, so that the effect of the latter on the flow regime is adequately accounted.

The algorithms used in the pressure-based solver compute the equations for the each variable in a sequential (or segregated) manner. Since the variables in the equations depend on each other, an iterative process needs to be followed to obtain the solution. Several algorithms are available to solve iteratively the pressure and velocity equations. These include SIMPLE, SIMPLER, PISO, Coupled, and Fractional Step (FSM) [70]. While SIMPLE, SIMPLER, PISO, and FSM are fully segregated algorithms, the Coupled scheme links the momentum and pressure equations, which are solved simultaneously. Typically, this method gives improved convergence rates compared to the segregated solvers,

although at a higher memory requirement [70]. Detailed information about these algorithms can be found in [70, 98], while Figure 3.8 summarises their structure.

In this work, the pressure-velocity coupling algorithms adopted are the PISO and the Coupled. The latter method is normally preferred, as it has been found that the convergence speed and stability of the simulation are generally improved compared to the PISO algorithm. The pressure values on the cell faces are interpolated using typically the pressure staggering option (PRESTO!), or the body force weighted option. The first uses the discrete continuity balance for a staggered control element about a given face to compute its pressure, while the body force weighted assumes that the normal gradient between pressure and body forces is constant. Other options include linear and second order interpolation, although they have not been used in this work. More details about the pressure interpolation schemes available can be found in [70].

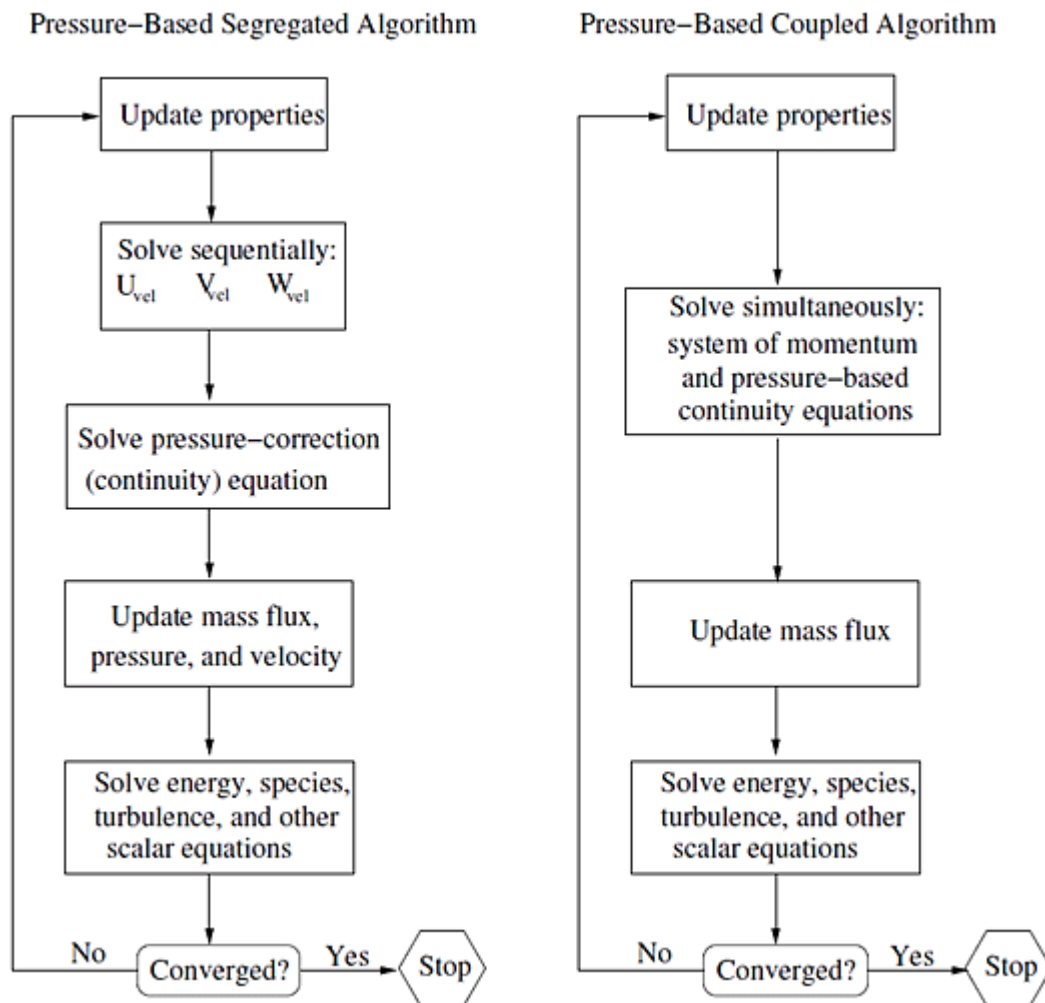


Figure 3.8 – Pressure-based solution algorithms [70]

4 Validation: KIT Bearing Chamber

4.1 Introduction

To assess the suitability and the key challenges of the VOF approach for bearing chamber simulations, a numerical model (“baseline model”) representing a simplified bearing chamber geometry is set up. The model reproduces the experimental layout and operating conditions reported in the work of Kurz et al. [10] on the KIT bearing chamber test rig. These experimental activities were conducted to characterise the oil wall film thickness distributions and scavenge efficiencies for different bearing chamber configurations and operating conditions. The work reported in [10] provides data on the oil film thickness distributions at various locations of the bearing chamber annulus, as well as information on the oil split between the vent and scavenge pipes. The data is used in this thesis to evaluate the accuracy of the numerical approaches tested. As discussed in Section 2.3.4, the most challenging conditions for the Volume of Fluid (VOF) method are those in which gas shear has a strong effect in driving the liquid film. Therefore, the chosen reference operating condition is the highest shaft rotational speed available from the experimental data set [10].

After defining the computational grid requirements with the help of single phase simulations and guidelines based on previous works, the work presented in this chapter focuses on the interface reconstruction approach, aiming to find a method ensuring both numerical accuracy and acceptable simulation times. For this purpose, the High Resolution Interface Capturing (HRIC), Compressive, and Geometric reconstruction schemes, described in Chapter 3, have been evaluated. The focus moves onto the gas/liquid interface turbulence damping approaches available. For the first time in bearing chamber applications, different turbulence damping factor values of Egorov’s approach [80], as well as the Mathematically Closed Interface Treatment proposed by Hashmi [39], are tested and evaluated against the experimental data. The results highlight the relevant effect that the turbulence damping choice has on the predicted oil distribution, as well as on the oil split between the vent and scavenge flows.

This chapter includes the results of the thermal simulation performed on the KIT bearing chamber to assess the capability of heat transfer modelling with VOF. Guidelines

and recommendations are provided at the end of the chapter to help setting-up bearing chamber models with this method.

4.2 Set-up

4.2.1 Bearing Chamber Geometry

Compared to the other configurations available in the reference experiment of Kurz et al. [10], the one selected for this work is the simplest geometry. The vent pipe is not protruding in the bearing chamber and the scavenge is a simple hole at the bottom of the annulus, i.e. without a ramp layout. The configuration is known as ‘baseline’ in the work of Kurz et al. [10]. The bearing chamber has a rectangular cross-section. Cylindrical vent and scavenge pipes having the same diameter are located on its centreline, as shown in Figure 4.1(a). The shaft crosses the whole axial length of the bearing chamber and is supported by roller bearings on one side. On the opposite side, a labyrinth seal covers the gap between the shaft and the static wall. In the numerical model, the seal is represented as a thin annular channel through which air is injected, enabling the build-up of the air flow profile and swirl velocity. Oil enters the domain as a continuous film through a ring covering the location of the bearings, as shown in Figure 4.1(b), following the guidelines of Krug [92].

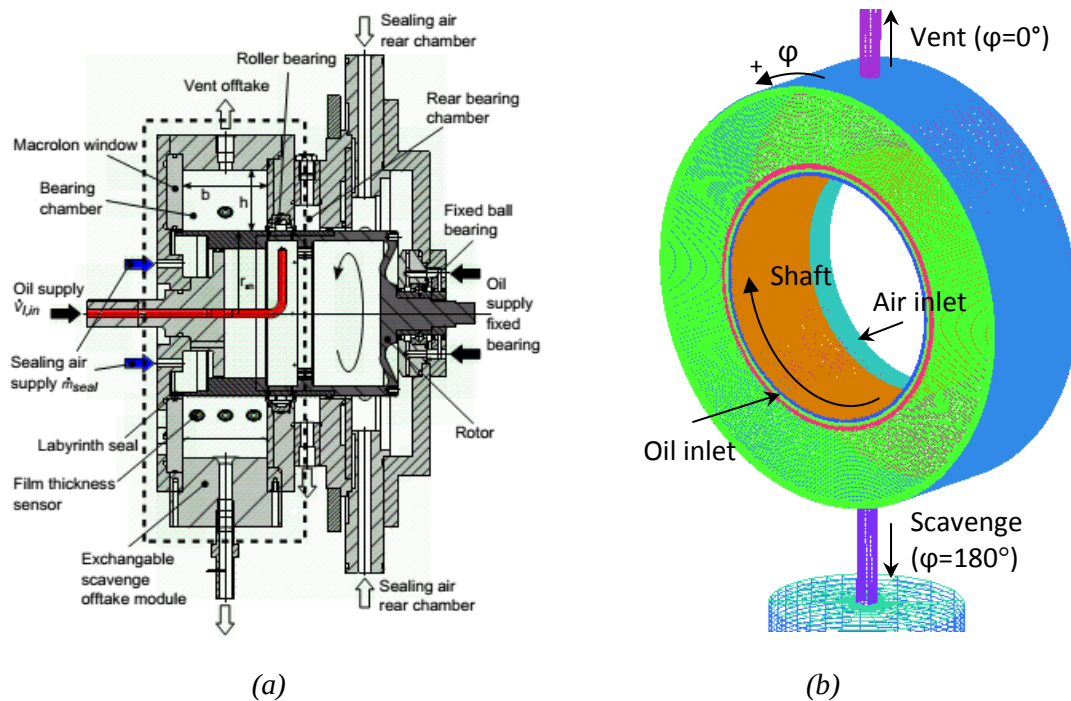


Figure 4.1 – KIT bearing chamber experimental setup [10] (a), and (b) corresponding computational domain

The numerical model has also a 'tank' located at the bottom of the scavenge pipe. The tank is constantly filled with oil to ensure that this is the only phase leaving the domain through its bottom outlet, following the pump model introduced by Robinson et al. [125] at Nottingham. The bearing chamber dimensions are summarised in Table 4.1.

Table 4.1 – KIT bearing chamber dimensions

Shaft diameter	ϕ_{sh}	128	mm
Chamber axial length	b	66	mm
Outer (rim) wall diameter	ϕ_{rw}	222	mm
Vent pipe diameter	ϕ_{vt}	10	mm
Scavenge pipe diameter	ϕ_{sc}	10	mm
Air inlet channel thickness*	$h_{a,in}$	0.7	mm
Oil inlet channel thickness*	$h_{o,in}$	2	mm

* In computational domain

4.2.2 Meshes

The computational grids adopted in the simulations have been developed as ANSYS ICEM structured grids. Typically, the mesh is refined near the chamber outer wall to resolve the oil film adequately. In line with the indications of Tkaczyk [43], to capture the velocity profiles of the liquid film the requirement has been set to have at least six cells through its thickness. Given a minimum observed film thickness of about 200 μm [10], this

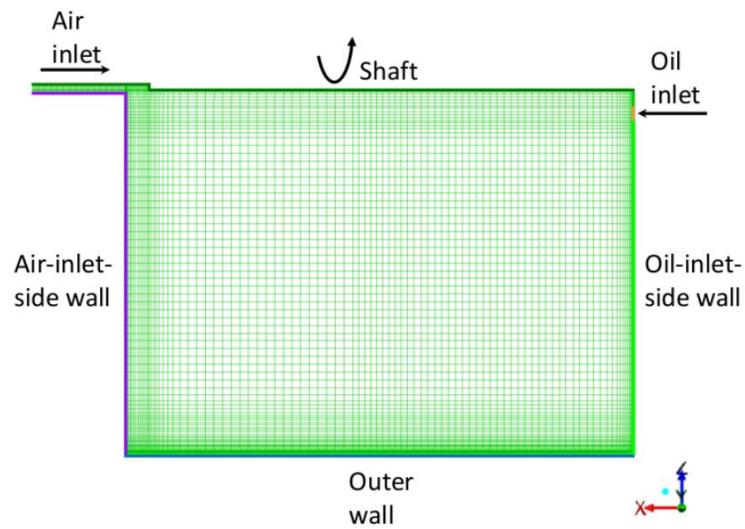


Figure 4.2 – Cross-section of KIT bearing chamber mesh

has led to a baseline mesh with 24 cells distributed radially in the region comprised within 2.3 mm from the outer wall, with the centroid of the first cell at 18.4 μm from the wall (first node spacing of 36.7 μm).

Figure 3.3 shows a cross-section of the baseline mesh in the plane orthogonal to the vent and scavenge pipes. The mesh defined here has 2.65 million cells, and is the one adopted for all the simulations in this chapter unless otherwise stated. Different mesh resolutions have been tested both in the near-wall and the main annulus regions, to ensure that the solution is independent from the level of refinement. An air-only, single phase mesh independence study relied on meshes having 680,000, 2 million, 2.6 million and 2.8 million cells. The results of this study are presented in Section 4.3. Further studies on the mesh independence of the oil film used meshes with 1.7, 2.7, 3.8 and 8 million cells instead. Their outcome is reported in Section 4.4.3.

4.2.3 Operating Conditions

The chosen shaft rotational speed for the simulations is 15,000 rpm. This value is the maximum available from the experimental set-up, and has been selected as reference in order to simulate the condition in which the air shear on the oil has its strongest effect, that is the most challenging condition to reproduce in VOF. The scavenge ratio, defined as

$$SR = \frac{\dot{V}_{sc}}{\dot{V}_{o,in}} \quad (4.1)$$

where $\dot{V}_{sc}$ is the volumetric flow extracted by the scavenge pump, and $\dot{V}_{o,in}$ the volumetric oil flow injected through the bearing, is set to 4 as in the reference experiment.

The oil inlet flow is 25.81 g/s (100 l/h). The bearing chamber is pressurised to 2.7 bar, with a sealing air flow of 10 g/s. The experiment [10] was conducted with constant air and oil inlet temperatures of 373 K. Table 4.2 summarises the operating conditions and fluid properties.

4.2.4 Boundary Conditions

With reference to Figure 3.3, the shaft is set as a moving wall, with a non-slip shear condition; all the other walls are stationary. Air and oil are both injected with a mass flow inlet boundary condition and a swirl angle $\gamma = 50^\circ$ between the velocity vector and the axial direction to match the shaft rotational speed, following the guidelines of Peduto [41].

Table 4.2 – Operating conditions and fluid properties

Shaft rotational speed	N_{sh}	15000	rpm
Air inlet temperature	$T_{a,in}$	373	K
Air inlet mass flow	$\dot{m}_{a,in}$	10	g/s
Oil inlet temperature	$T_{o,in}$	373	K
Oil inlet flow	$\dot{m}_{o,in}$	25.82	g/s
	$\dot{V}_{o,in}$	100	l/h
Chamber absolute pressure	P_{ch}	2.7	bar
Pump scavenge ratio	SR	4	[-]
Material properties			
Air density	ρ_a	2.52	kg/m ³
Air viscosity	μ_a	$2.21 \cdot 10^{-5}$	kg/(m·s)
Oil density	ρ_o	929.5	kg/m ³
Oil viscosity	μ_o	$4.83 \cdot 10^{-3}$	kg/(m·s)
Oil-air surface tension	σ_{o-a}	$2.45 \cdot 10^{-2}$	N/m

The oil sump outlet has a constant negative velocity boundary condition to match the pump scavenge ratio, while the vent pipe outlet at the top has a pressure boundary set to the chamber operating pressure value. Analogously with the work of Krug [92], who derived the air inlet turbulent intensity I from

$$I = 0.16 Re_h^{-1/8} \quad (4.2)$$

the latter value has been set for the air inlet to 6%, with a hydraulic diameter of 1.4 mm.

The oil inlet turbulent intensity and hydraulic diameter are 0% and 3.0 mm respectively, while the air outlet boundary in the vent pipe has a backflow turbulent intensity of 4% and a hydraulic diameter of 10 mm.

The effect of the surface tension between the oil and the air is accounted for by using the Continuous Surface Force model of Brackbill et al. [126].

4.2.5 Turbulence Models

Near-Wall Treatment

As discussed in Section 3.4.5, the SST k - ω turbulence model has been selected for the baseline simulation to ensure appropriate modelling of the near-wall boundary layer for

the interval of y^+ values found in the bearing chamber. These can be as low as 0.3 on static locations such as the chamber side wall, while the moving walls, namely the shaft surface, give an average of 66.0. Preliminary single-phase tests on the bearing chamber with y^+ values of 3 and 70 near the shaft gave a wall shear stress discrepancy of about 0.9%, confirming the suitability of the SST $k-\omega$ model for the simulations carried out in this work. All the simulations reported in this chapter have been performed using this model, except those implementing the gas/liquid interface treatment proposed by Hashmi [39]. The cases based on this method rely on the RNG $k-\epsilon$ turbulence model, as the source term was developed and tested by Hashmi [39] using the latter. The ‘Enhanced wall treatment’ near-wall formulation has been chosen in this case.

Free Surface Treatment

As discussed in Section 3.5, bearing chamber simulations performed with the VOF method require damping of the turbulent momentum transfer at the gas-liquid interface. The work presented in this chapter evaluates Egorov’s turbulence damping [80] and Hashmi’s Mathematically Closed Interface Treatment (MCIT) [39]. The two methods are described in Sections 3.5.2 and 3.5.3 respectively.

As Egorov’s turbulence damping [80] is dependent on the choice of the scaling factor B , two cases have been set-up using respectively $B = 10$ and $B = 100$ to evaluate its impact on the oil distribution and split between the outlet pipes. The first value is by default the minimum factor required by the model [80], while $B = 100$ has been chosen following the guidelines of Tkaczyk [43], who suggests this value for the best match of the experimental gas and liquid velocity profiles in a stratified rectangular channel flow. The implementation of the MCIT does not require any scaling factor to be set instead.

4.2.6 Solver Settings

Equations

The solver relies generally on a bounded second order implicit transient formulation, with coupled pressure-velocity equations and body force weighted pressure discretization. Momentum, turbulent kinetic energy, and dissipation equations use second order upwind spatial discretization. This set-up covers most of the cases tested, except for few ones where different settings such as first order implicit transient formulation had to be adopted. The settings which differ from the described baseline set-up are clearly reported for each specific case in the following sections.

Time stepping

As a simulation start-up procedure, the time step size has been limited to 10^{-6} s for 1000 time steps before switching to a fixed 10^{-5} s. The latter value ensures for the baseline mesh a Courant number value of about 0.4 on the shaft surface and always less than unity in the main chamber annulus, i.e. where the oil film thickness is computed.

Interface Reconstruction

As one of the objectives of this work is to find a way to speed-up the simulation of long transients in bearing chamber VOF simulations, the influence of the interface computation method has been taken into account. Typically, these simulations rely on the Geometric Reconstruction method [23, 41, 92] because of its high accuracy [71]. One drawback of this method is the need to use an explicit formulation of the VOF equation (3.16), which limits the time step size that can be set in the solver due to the need to keep relatively low Courant number values in the whole domain. As the implicit formulation of the VOF equation would ensure simulation stability for higher Courant numbers than the explicit, both interface computation methods available for this formulation in ANSYS Fluent have been tested to assess their suitability. These are the Compressive interface reconstruction and the High Resolution Interface Capturing (HRIC). All three methods tested are evaluated in Section 4.4.1.

Initialisation

The flow initialisation has been performed starting from the air-only solution and patching a 0.5 mm oil layer on the chamber oil inlet-side and outer wall. This approach has been followed in order to avoid the simulation of the transient period required to fill a dry bearing chamber, in-line with the procedure proposed by Krug [92].

4.2.7 Film Thickness Averaging

The oil film thicknesses are computed as the distance between the iso-surface of the oil volume fraction $\alpha_o = 0.5$ and the chamber outer wall. To obtain an adequate comparison with the experimental measurements, the iso-surfaces of α_o are time-averaged over a sufficiently long interval, typically of the order of 0.5 seconds. Two time-averaging options have been evaluated:

- Internal data sampling and time-averaging from the CFD software;
- Manual data sampling and time-averaging with purpose-built Matlab tool.

The reason for this approach was the need to verify that the time-averaging option provided by the CFD software gives reliable results. Using a manual sampling frequency of 1 kHz, corresponding to the sampling frequency of the film thickness sensors in the experiment [38], the two approaches have proven equivalent. This highlights that the internal time-averaging of the iso-surfaces in the CFD software can be used to post-process the results, as well as that a sampling frequency of 1 kHz gives results equivalent to higher sampling frequencies (i.e. the CFD internal data sampling). The latter observation suggests that sampling frequencies of 1 kHz are sufficient to capture the oscillatory behaviour of the oil film, supporting the experimental choice.

In the reference experiment [10], oil film thicknesses are measured through capacitive sensors located at eight circumferential positions around the bearing chamber annulus, as illustrated in Figure 4.3. To perform a comparable analysis, the oil film thicknesses are computed at discrete locations also in the numerical simulation, and space-averaged over defined areas. For practical reasons, the space averaging areas in the post-processing tool have been chosen with a rectangular shape, while the sensors of the experiment are circular. To match the area covered by the sensors ($\varnothing_{exp} = 10 \text{ mm}$), the space-averaging regions are defined as squares with side length of 5 mm. Figure 4.4 shows the typical layout of the film thickness computation areas chosen for the simulation, as well as the position of the film thickness sensors of the reference experiment [10]. The ‘numerical’ sensors can be moved both axially and circumferentially inside the chamber, as well as changed in number, to better evaluate the spatial variations of the oil film thickness.

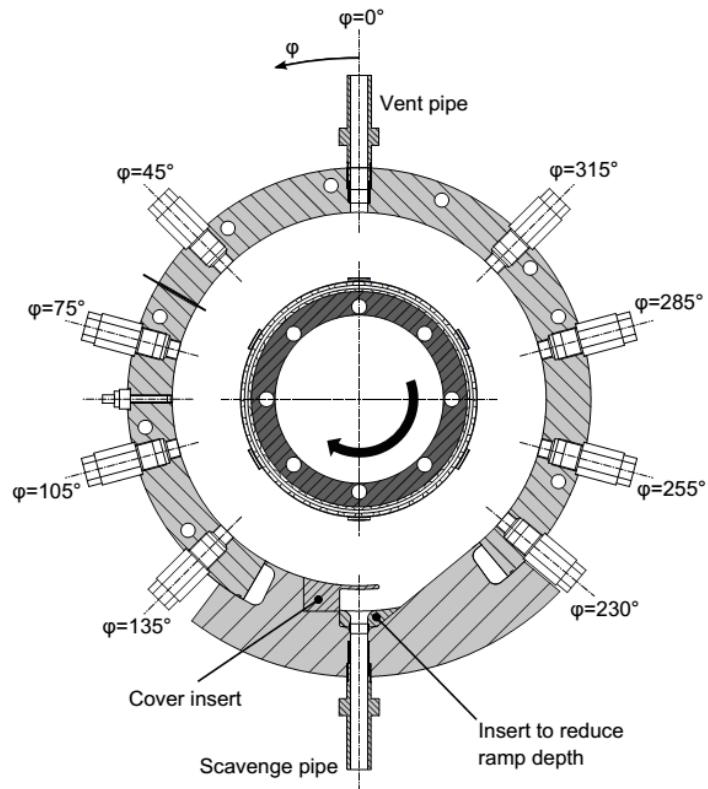


Figure 4.3 – Location of film thickness sensors in KIT bearing chamber [10]

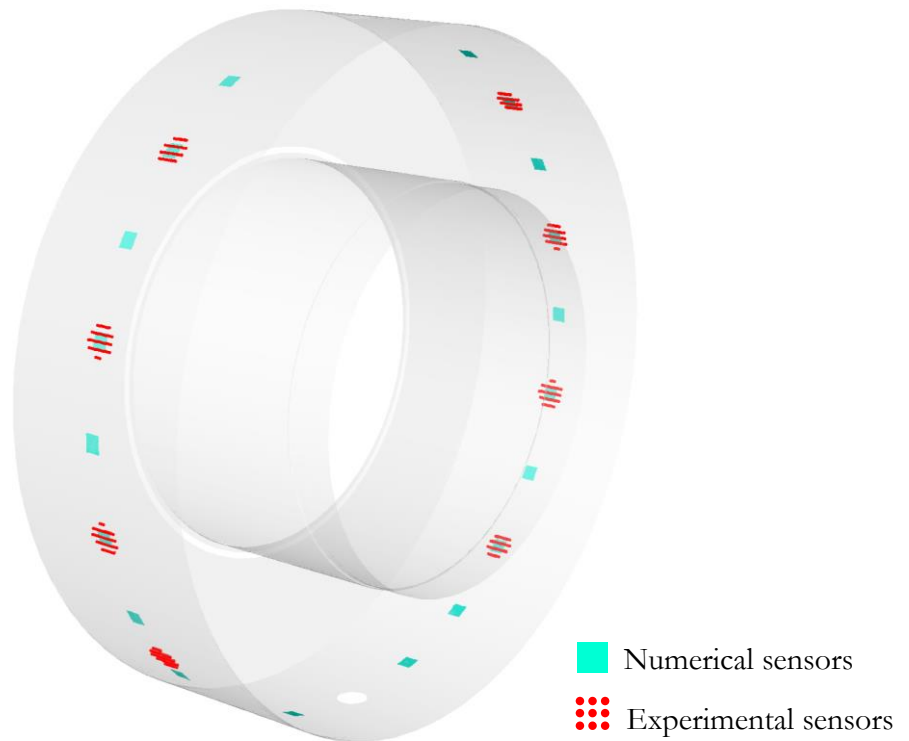


Figure 4.4 – Numerical and experimental oil film thickness measurement areas

4.2.8 Set-up Summary

Table 4.3 summarises the general two-phase flow simulation settings. The single phase, air-only simulations of Section 4.3 adopt the same boundary conditions reported here, with the oil inlet replaced by a stationary wall. They rely on a steady-state calculation with pseudo-transient under-relaxation, and adopt the SST k - ω turbulence model.

Table 4.3 – General two-phase simulation settings

Multiphase Model	Volume of Fluid (VOF)		
Turbulence Model	SST k - ω		RNG k - ϵ
Near-wall Formulation	-		Enhanced wall treatment
VOF Turbulence Damping	Egorov [80]		Hashmi's MCIT [39]
Damping Factor	10	100	-
Boundary Conditions	Oil	Mass flow inlet: $\dot{m}_{o,in} = 25.82 \text{ g/s}$ Inlet angle $\gamma = 50^\circ$	
	Air	Mass flow inlet: $\dot{m}_{a,in} = 10 \text{ g/s}$ Inlet angle $\gamma = 50^\circ$	
	Vent	Pressure outlet, $p = 1.7 \text{ bar}$	
	Sump	Velocity outlet $U_{out} = 0.013837 \text{ m/s}$ (chamber SR=4)	
	N shaft	15000 rpm	
Pressure-Velocity Coupling	Coupled		
Gradient	Least squares cell based		
Pressure Discretization	Body force weighted		
Momentum Discretization	Second order upwind		
Turbulent Kinetic Energy Discretization			
Dissipation/Specific Diss.			
Body Force Formulation	Implicit body force		
Volume Fraction Formulation	Explicit		Implicit
Interface Computation	Geometric reconstruction		Modified HRIC Compressive
Transient Formulation	First order implicit		Bounded second order implicit
Time step Size	$2 \cdot 10^{-6} \text{ s}$		10^{-5} s
Mesh Size (Cells)	$2.65 \cdot 10^6$		
Initialisation	0.5 mm oil film on bearing-side and rim wall Single phase air flow established		

4.3 Single-Phase Results

As stated in Section 4.2.2, a single-phase mesh independence study has been performed to understand the minimum grid requirements to capture the typical air flow behaviour of the bearing chamber. The underlying principle is to define a mesh capable of describing correctly the core air flow, and successively apply the best practice to resolve the oil film region in the two-phase flow simulation. This study considers the secondary air flows, namely in the planes orthogonal to the shaft surface (i.e. the radial and axial components of air velocity). The radial and axial velocity profiles are compared in a reference plane, located at $\varphi = 270^\circ$ as shown in Figure 4.5 (a). The comparison is done by plotting the profiles along two radial lines and one axial line of the chamber section, illustrated in Figure 4.5 (b). Four meshes with different refinements in the chamber annulus region, sized 680,000, 2 million, 2.6 million, and 2.8 million cells, have been tested.

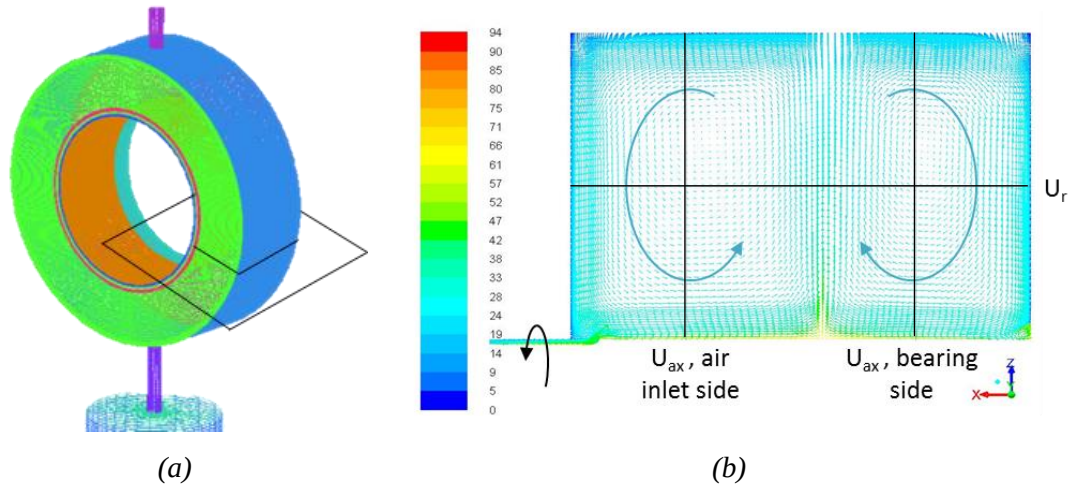


Figure 4.5 – Location of section plane at $\varphi=270^\circ$ for secondary flows mesh independence study (a); Velocity vectors [m/s] in section plane for fine mesh and location of velocity plots (b)

Indications on the expected secondary flow regime are found in the work of Gorse et al. [18], who performed secondary air flows measurements on a bearing chamber which has a similar geometry to the one in this work. The chamber of Gorse et al. [18] is 33% shorter than the one presented here, with the shaft rotating at a 13% lower speed; all the other dimensions and operating conditions are the same as in this case. To characterise the secondary air flow conditions, the velocity ratio R of Equation (2.6) is used. Gorse et al.

[18] found that for $R > 5$ the secondary air flow is driven by the shaft rotational speed (Rotational Speed Driven Mode), which prevails over the axial momentum contribution given by the sealing air flow. This flow condition is characterised by two counter-rotating vortices, whereas at lower shaft speeds the sealing air flow tends to generate a single vortex. Computing the velocity ratio for the operating conditions simulated in this work, $R = 7.09$; hence, two counter-rotating vortices are expected.

The results show that the coarse mesh is not sufficient to resolve the two counter-rotating vortices. Finer meshes can identify the vortices instead, as shown by the overlapping 2, 2.6 and 2.8 million cells plots in Figure 4.6. Refinements above 2.8 million cells did not give any further change in the results. The corresponding cell size for mesh independence is of the order of 2 mm in the main gas flow path, i.e. far from the oil film regions on the walls. The latter guideline for the mesh in the core air flow should be applied with caution, as the successive two-phase simulations have shown that the two-vortex structure will be disrupted by the air-oil interaction. The air flow field in this case becomes characterised by several small, unsteady vortices, as shown in Figure 4.7. Hence, the previously defined mesh requirements may no longer be sufficient to characterise the new vortex structures. Considering the latter change in the structure of the air vortices, it can be stated that the distribution of oil inside the chamber cannot be determined *a priori* by simply looking at the shape and direction of the ‘air-only’ vortices. In fact, this distribution will depend on the interaction with the air flows, but these will be influenced in turn by the amount and distribution of oil inside the chamber.

Based on the considerations above and keeping in mind the different scenario of a two-phase simulation, a core mesh size of 2 mm has been selected as starting point for the successive VOF simulations. Additional mesh independence studies have been carried out on the multiphase case. The results are presented in Section 4.4.3.

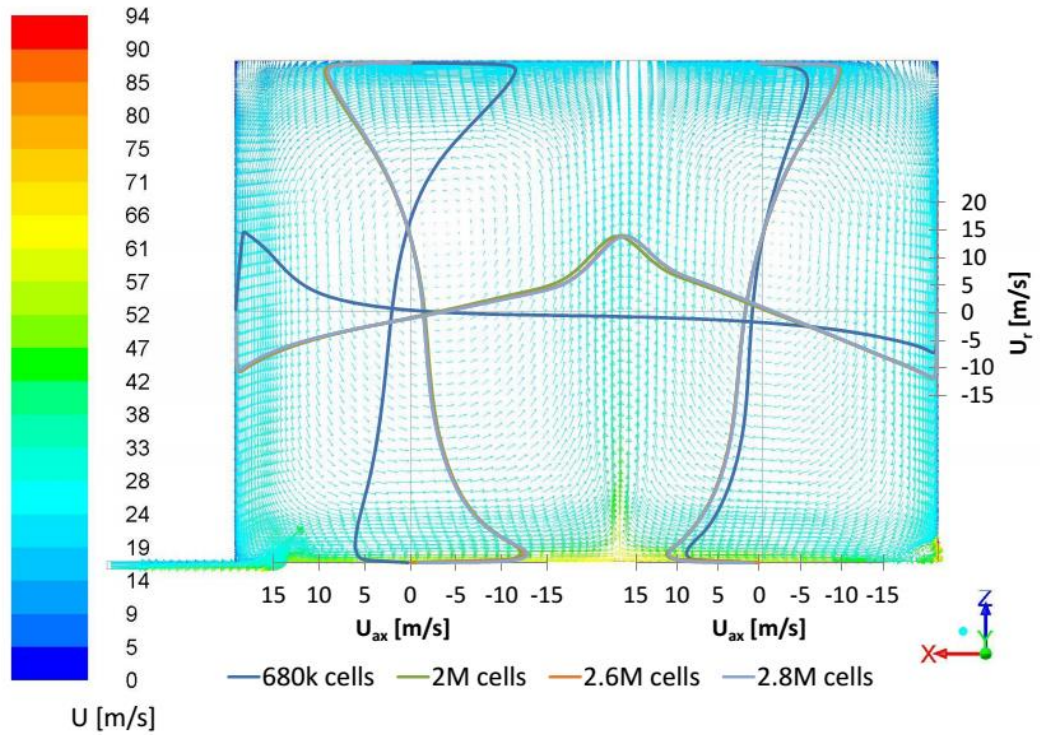


Figure 4.6 – Air velocity vectors and plots in section plane $\phi = 270^\circ$; velocity vectors for 2.8 million cells mesh

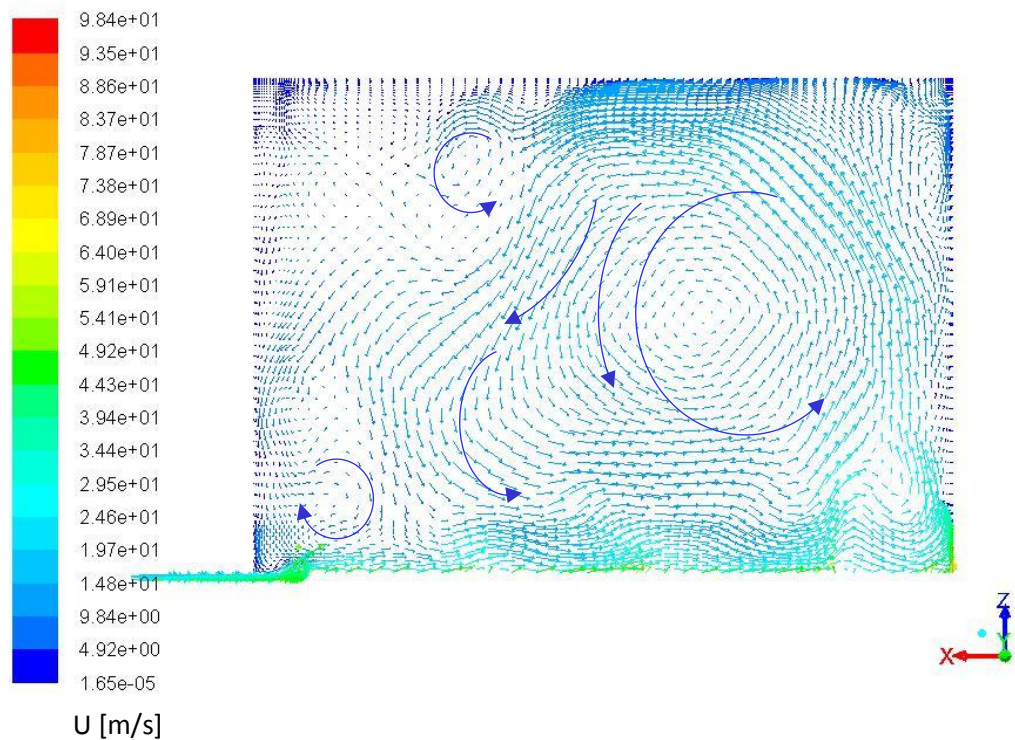


Figure 4.7 – Example of instantaneous air velocity vectors in section plane $\phi = 270^\circ$ for two-phase VOF simulation. Mesh size: 2.65 million cells

4.4 Two-Phase Results

4.4.1 Evaluation of Interface Computation Methods

Out of the three VOF gas/liquid interface computation methods listed in Table 4.3, the Geometric reconstruction has been soon found unsuitable for the mesh adopted because of its time step size restrictions; this is because of the explicit formulation required by this method in VOF (see Section 3.3.4). Test runs have identified in $2 \cdot 10^{-6}$ seconds the upper limit of the time step needed to avoid convergence issues in the simulation. Time steps of this order of magnitude are excessively small for the requirement of simulating physical times of multiple seconds on a problem of this scale. Because of the limited range of adjustment of the mesh (which is constrained by the need to keep thin cell layers on the bearing chamber walls), it was deemed more practical to move towards an implicit VOF formulation.

The ANSYS Fluent modified HRIC scheme was the first to be tested combined with the implicit formulation. This model highlighted a tendency to numerical diffusion at the interface, leading to dry-out of the oil film at the walls, in disagreement with the experimental observations. The latter behaviour is clearly identifiable in the plots of Figure 4.8. This characteristic is not found when using the Compressive interface reconstruction instead. As shown in Figure 4.9, the minimum oil film thickness tends to remain constant at about 0.15 millimetres, thus not showing a numerical dry-out as in the modified HRIC.

The low numerical diffusivity found in the Compressive scheme confirms the recommendation over the choice of VOF interface computation schemes made by Seo in the ANSYS best practice guidelines [71, 127]. His report highlights the tendency of the modified HRIC scheme to give numerical diffusion at the interface, and suggests using the Compressive scheme for an optimal compromise between interface computation accuracy and simulation speed.

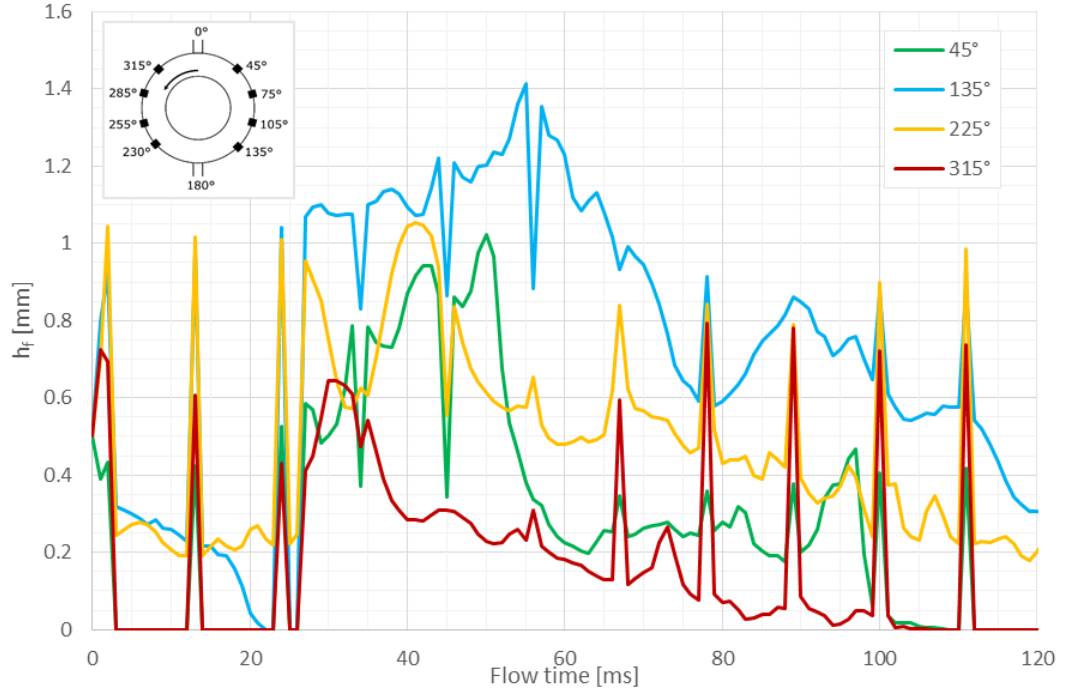


Figure 4.8 – Time evolution of oil film thickness, modified HRIC scheme. Plots at different circumferential locations of the bearing chamber outer wall

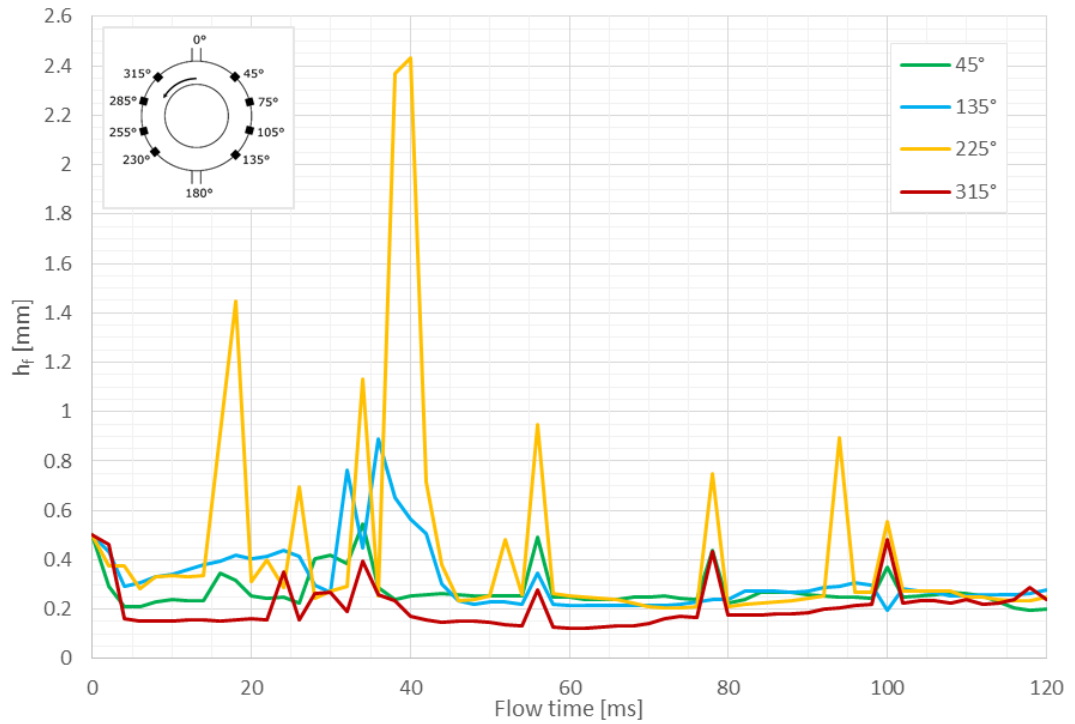


Figure 4.9 – Time evolution of oil film thickness, Compressive reconstruction scheme. Plots at different circumferential locations of the bearing chamber outer wall

Encouraging results on the application of the Compressive scheme are also presented by Peduto in his PhD thesis [9], where good qualitative and quantitative agreement with

experimental data is achieved in the simulation of droplet splashing. Other examples on the application of this scheme can be found in [34, 128].

For the reasons above, the Compressive interface reconstruction scheme has been selected as the default approach for the bearing chamber simulations in this work. Compared to the more expensive Geometric reconstruction, it has enabled the reduction of the simulation times from about 80 to 12 days, mainly thanks to the increase of the time step size from $2 \cdot 10^{-6}$ seconds with the Geometric reconstruction to 10^{-5} seconds here. The Compressive scheme applied to a 2.65 million cells mesh has led to the simulation of 1 physical second in 140 hours on 96 cores (Intel Sandybridge E5-2670 2.6GHz 2x8-core nodes), equivalent to a CPU cost of 13,440 CPUh/s. For comparison, the projected time to compute one second of flow with the Geometric reconstruction scheme is 1,840 hours, using a 1.35 million cells mesh on 48 cores, keeping the same cells-per-core ratio of the corresponding Compressive case mesh. Therefore, the CPU cost of the Geometric reconstruction scheme has been estimated at 88,320 CPUh/s. Due to the prohibitive computational time required, the simulation adopting the latter scheme has not been carried out to the point where a flow stationary state is achieved, and its results are not reported here.

4.4.2 Baseline Case

Introduction

The results presented in this section are the outcome of the first in the series of transient simulations that achieved the flow stationary-state. This simulation has been defined therefore as ‘baseline’ for comparison with all the successive cases performed in this work. With reference to Table 4.3, the baseline case relies on the SST k- ω turbulence model with Egorov’s turbulence damping [80]. The damping factor value is 100, and the implicit VOF formulation with Compressive interface reconstruction is adopted. All the remaining settings are univocally listed in Table 4.3.

The choice of using the implicit VOF formulation has enabled the use of a time step size of 10^{-5} seconds for the given 2.65 million cells mesh. This corresponds to a maximum Courant number value of 17.3 at the vent pipe inlet, where flow velocities of the order of 100 m/s and cell sizes lower than 0.1 mm create a critical region for the flow solution. The zone affected by the high Courant number value is only in proximity of the vent pipe, hence the oil film behaviour in the domain is assumed to be undisturbed by this region. Convergence in about 4 iterations per time step has been maintained throughout the whole

simulation, with a maximum residual criterion of 10^{-3} for all the computed variables. The relatively fast time advancement has enabled to compute several seconds of flow on a much finer mesh than the ones found in previous full bearing chamber simulations [92], improving the resolution while keeping acceptable computational times. This is a significant improvement over the previous activities with the explicit VOF formulation, such as those of Adeniyi [47], where the flow was modelled only for about 150 milliseconds, or those of Krug [92], which achieved steady state but relied on a coarse mesh (621,500 cells) to use sufficiently large time steps.

Oil Residence Volume

The oil residence volume is defined as the amount of oil contained in the bearing chamber annulus, i.e. excluding the outlet pipes and the bottom tank, at any given time. Although experimental data is not available for this quantity in the selected experiment, it is particularly important in the numerical simulations as its time evolution indicates the achievement of the flow stationary state in the system. The latter is defined as the condition at which given quantities oscillate periodically around a constant average value.

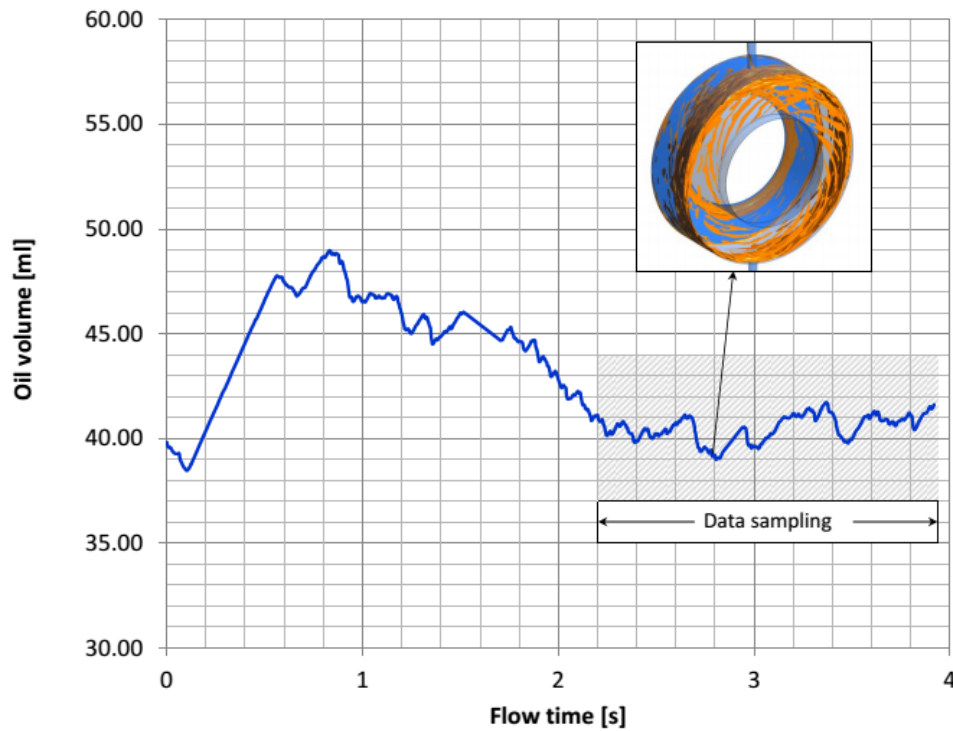


Figure 4.10 – Oil residence volume against flow time in KIT bearing chamber annulus, baseline case

Figure 4.10 plots the time evolution of the oil residence volume, starting from the initial state of a patched oil film on the bearing-side and outer walls, until the achievement

of the stationary state. The latter condition is achieved after about 2.2 seconds of flow time, as the oil volume starts oscillating with a nearly periodic pattern. The simulation has been run for further 1.8 seconds to assess the stability of the oil volume pattern and to collect time-averaged flow quantities. This interval is marked as ‘data sampling’ region in the plot.

In the ‘data sampling’ region the oil residence volume stabilises at an average of 40.5 millilitres, with oscillations of ± 2 ml. This behaviour corresponds to a rather smooth oil film in the bearing chamber, with small variations in the amount accumulating in the annulus. This is typical of shear-driven liquid flows, as the air flow is sufficiently strong to level-out the accumulations of oil which may start to form. The computed flow condition is therefore qualitatively aligned with the experimental observations reported in [10].

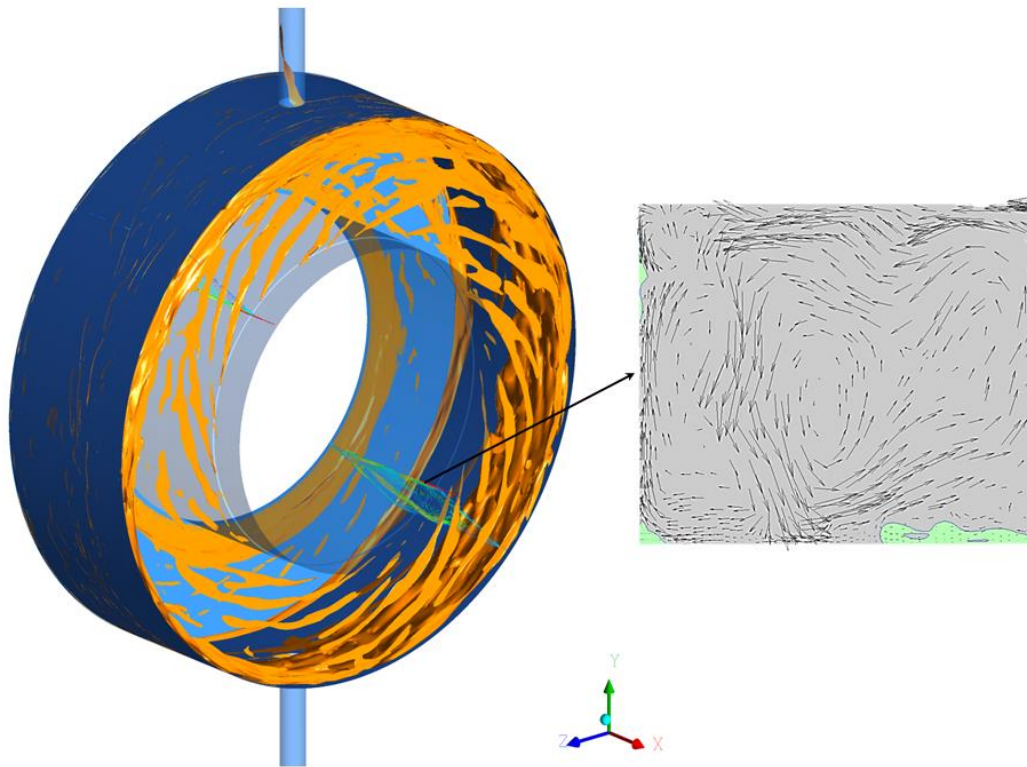


Figure 4.11 – Iso-surface of oil volume fraction in baseline case , $\alpha_o = 0.3$ at $t = 2.82$ s and in-plane velocity vectors at $\varphi = 135^\circ$

Film Thickness Distribution

From a qualitative overview of the oil distribution, it is observed that the air flow field tends to push the oil towards the air inlet (or ‘front’) side of the chamber. This is clearly visible in Figure 4.11, which shows an accumulation of oil consistent with the velocity

vectors of the secondary air flow. This result does not match the secondary vortices calculated in the air-only simulations and shown in Section 4.3. In this case, the air-oil interaction alters the morphology of the secondary flows, breaking the two counter-rotating vortices into multiple, smaller, unsteady vortices. Driven by the sealing air flow, the vortices tend to push the oil towards the front wall, leading to the accumulation shown in the figure.

As comprehensive information on the axial film distribution is not available from the reference experimental set [10], it is still premature to state that the prediction of an oil accumulation on one side of the chamber, driven by the sealing air momentum, is an accurate representation of the real flow. Additional data covering this quantity at several circumferential locations would help to confirm the validity of this statement.

In the circumferential direction, the time-averaged oil film thicknesses have been computed following the procedure described in Section 4.2.7. Figure 4.12 shows the results for the measuring areas shown in Figure 4.4, compared with the reference experimental data [10]. A clear under-prediction of the oil film thickness is found in the numerical model. The main flow features captured are the thinning of the oil film from the top towards the bottom of the co-current side of the annulus (195° – 330°), as well as the peak in oil film

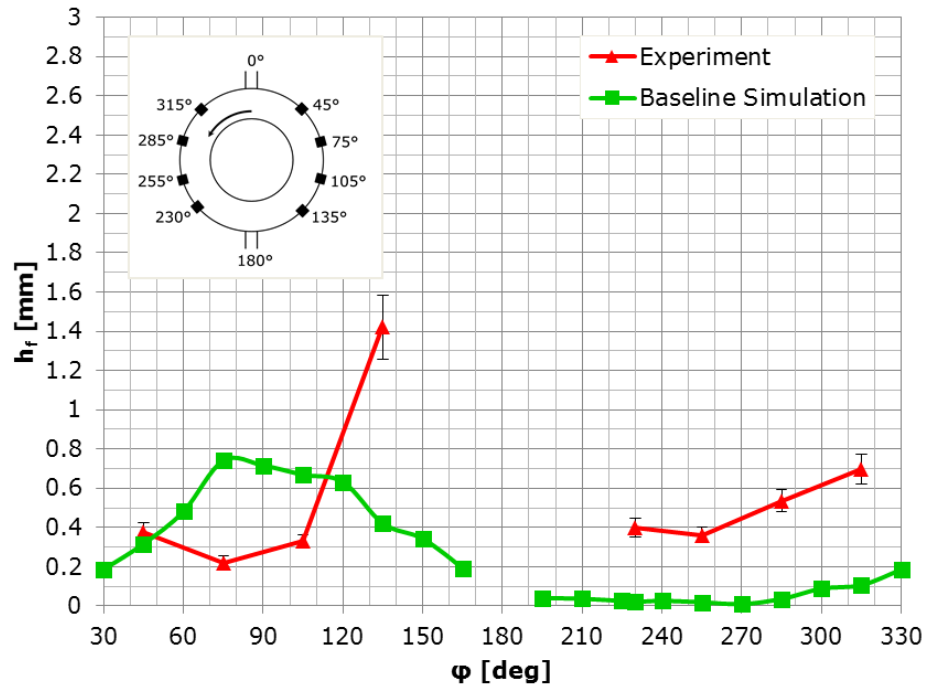


Figure 4.12 – Time-averaged circumferential oil film thickness distributions and reference experimental data [10]. Numerical distributions time-averaged over 0.2 s interval.
Baseline case

thickness in the counter-current side. However, the absolute film thickness values, as well as the position of the peak, are shifted compared to the experimental observations. The resulting time- and space-averaged film thickness calculated on the iso-surfaces of $\alpha_o = 0.5$ is 0.25 mm, against an experimental value of 0.54 mm. Post-processing tests included moving the locations of the sensors either axially or circumferentially. Although moving the sensors axially towards the air-inlet side wall by 2 or 4 mm gives higher film thicknesses, the trends remain basically unchanged, confirming the shortcomings in the accuracy of the oil film thickness distribution predictions in this baseline simulation.

Scavenge Efficiency

Having defined the scavenge efficiency as

$$\eta_{sc} = \frac{\dot{m}_{o,sc}}{\dot{m}_{o,in}} \quad (4.3)$$

(the ratio between the scavenged and injected oil mass flow rates), the calculated scavenge efficiency is lower than the experimental observations. In fact, a significant amount of oil tends to entrain with the air flow and to leave the domain through the vent pipe, as shown in the oil flow rates plot of Figure 4.13. Although this characteristic of the oil flow was also observed in the experiment of Kurz et al. [10], who found that about 62% of the oil injected leaves the chamber through the vent pipe, this phenomenon is over-predicted by the numerical model, estimating that 90% of it enters the vent pipe. The computed scavenge efficiency is therefore of the order of 10%, against an experimental value of 38%.

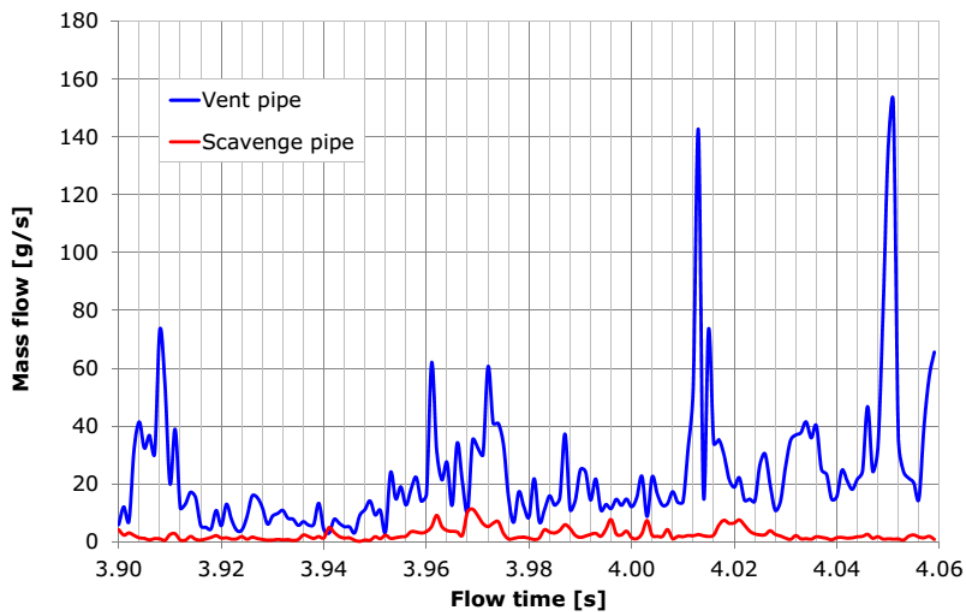


Figure 4.13 – Time evolution of vent and scavenge pipe oil mass flow rates, baseline case

Discussion of the Results

The results reported in this section show that the Volume of Fluid set-up (using the present baseline settings) is not sufficiently accurate for the prediction of the oil film distribution on the bearing chamber wall, as well as for estimating the scavenge efficiency correctly. As air shear is the main driver defining the oil flow characteristics and distribution [129], the results suggest potential inaccuracies in the modelling of the air-oil interaction. These may be due to:

1. Inaccuracies in the Compressive interface reconstruction method;
2. Insufficient mesh resolution in the oil film region;
3. Inadequacy of the turbulence damping model as formulated.

Having ruled out the choice of the Compressive interface reconstruction as a reason for inaccuracies, based on the discussion in Section 4.4.1, the two areas where further investigations are needed are 2. and 3. Sections 4.4.3 and 4.4.4 address these two points.

4.4.3 Oil Films Mesh Independence Study

Settings

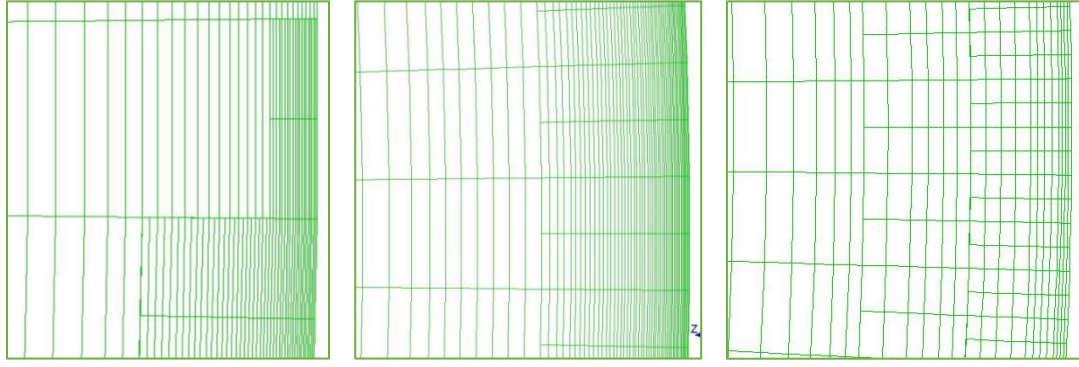
Different mesh refinements have been performed to evaluate whether the grid size, particularly in the axial and circumferential directions, could have any impact on the oil film thickness distribution. For this purpose, three cases have been created. All of them have an additional mesh refinement with hanging nodes in the oil film region near the outer wall, as hereby listed:

- (a) 1 level of refinement in the 2 mm near-outer-wall layer in $90^\circ < \varphi < 180^\circ$;
- (b) 1 level of refinement in the 2 mm near-outer-wall layer in the whole chamber annulus;
- (c) 2 levels of refinement in the 2 mm near-outer-wall layer, 1 level of refinement in the 2-4 mm layer, in the whole chamber annulus.

Cases (a), (b), (c) are illustrated in Figure 4.14.

Convergence

All the cases have used as starting point the final flow field of the baseline case (see Section 4.4.2) at $t=3.899$ s. Due to time and computational resources constraints (the largest mesh has a size of 8 million cells), all the cases have been run for a fixed time interval of 0.1 s.



(a)

(b)

(c)

Figure 4.14 – Near-wall mesh refinement approach

Results

The results of the three cases, compared to baseline, are reported in Figure 4.15. From the comparison of different refinement levels in the oil film region, it can be stated that the mesh changes have only a small effect on the film thickness trends in the time interval considered, except for case (a) where the non-axisymmetric refinement affects the oil distribution in the non-refined regions. The results therefore suggest that further mesh refinement is unlikely to improve the accuracy of the solution; therefore, the discrepancy with the experimental data must depend on other aspects of modelling. For this reason, the investigations have moved further onto the analysis of the effect of the turbulence damping approach.

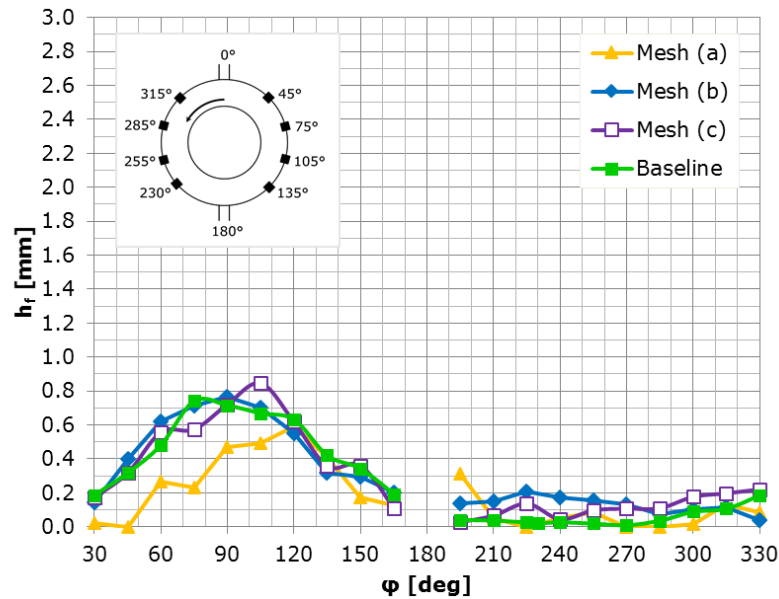


Figure 4.15 – Mesh independence study for oil film thickness circumferential distribution. Values time-averaged over 0.1 s.

It could be argued that the simulations may not have been run for a sufficient time, but case (a) shows that noticeable changes in the oil thickness distribution are already identifiable in the given time interval. The flow distribution has been able to adapt to a change in conditions and stabilise in its new state over that period, thus it is concluded that the simulations have been run for a sufficient time. This was also found true for cases (b) and (c).

4.4.4 Effect of Turbulence Damping

Introduction

The work presented in this section evaluates the effect of different turbulence damping choices on the behaviour of the bearing chamber oil flows. The section compares the results obtained with two different turbulence damping factors in Egorov’s approach [80], as well as with Hashmi’s Mathematically Closed Interface Treatment (MCIT) [39]. The main purpose is to determine whether the selection of the damping approach can have an impact on the accuracy of the results, highlighting a shortcoming in VOF for this class of problems, and if there is an opportunity to improve the matching with the experimental data compared to the baseline case discussed in Section 4.4.2.

Setup

In addition to the case of Section 4.4.2, two other simulations have been performed. With reference to Table 4.3, these simulations adopt two different turbulence models and turbulence damping approaches. The first simulation relies on Egorov’s method [80] and tests a turbulence damping factor value $B = 10$, as opposed to the baseline case $B = 100$. The chosen value of B in the new simulation is the default factor given by Egorov in his report [80]. This value has been adopted also by Krug [92] in his bearing chamber simulations, thus it has been considered a sensible value to be tested. For practical reasons, the baseline case and the new case will be named in this section respectively ‘TD100’ and ‘TD10’. The second additional simulation tests Hashmi’s MCIT method [39]. As discussed in Section 4.2.5, the MCIT in this simulation relies on the k - ϵ model. The simulation settings are the same as in the baseline case, thus the implicit VOF formulation with Compressive interface reconstruction is adopted for all cases. All the remaining settings are univocally listed in Table 4.3. Table 4.4 summarises the cases compared in this section.

The initialisation of the ‘TD100’ and ‘MCIT’ cases uses the methodology presented in Section 4.2.6, i.e. the flow field is computed in a preliminary air-only simulation and an

Table 4.4 – Turbulence damping cases tested

Case	Turbulence Model	Damping Method	B
TD100	SST k- ω	Egorov [80]	100
TD10	SST k- ω	Egorov [80]	10
MCIT	RNG k- ϵ	MCIT [39]	-

oil film layer of 0.5 mm is patched on the air inlet-side and outer walls. The ‘TD10’ case starts from the ‘TD100’ solution instead, as the original purpose of the simulation was to demonstrate that a change in turbulence damping factor leads to a full re-distribution of an already steady oil film.

Oil Residence Volume

To compare the time evolution of the oil residence volumes for the three cases investigated, a characteristic time scale has been defined as follows,

$$\tau = \frac{\pi \phi_{ch}}{\bar{U}_{\perp,a}} \quad (4.4)$$

where $\bar{U}_{\perp,a}$ is the volume-weighted average of the tangential air velocity in the bearing chamber annulus. The characteristic time scale therefore represents the average time required by a particle of air to complete one revolution in the annulus. The plot of Figure 4.16 clearly shows how the characteristic time scales and the oil residence volume are connected and affected by different turbulence damping approaches. High values of τ correspond to a low average air tangential velocity in the bearing chamber annulus. The latter behaviour is found in case TD10. This case shows also a large discrepancy, when compared with cases TD100 and MCIT, in terms of oil residence volume magnitude and amplitude of its oscillations. To explain these differences in behaviour, the balance between gravity and viscous forces in the liquid has been considered. The latter balance is quantified with the gravity parameter λ , computed as defined by Kay et al. [50],

$$\lambda = \frac{\rho_l g h_0^2}{\mu_l U_0} \quad (4.5)$$

where ρ_l is the liquid phase density, h_0 the reference film height, μ_l the liquid dynamic viscosity, and U_0 the reference film speed. The computed values of λ and the reference (average) film quantities for the three cases tested are summarised in Table 4.5.

The TD10 case is characterised by a near-balance between the viscous and gravity forces, having a λ value close to unity. Kay et al. [50] state that the combination of $\lambda \approx 1$ and low volume of oil filling the annulus (i.e. $F < 20\%$) gives a shock solution: surface

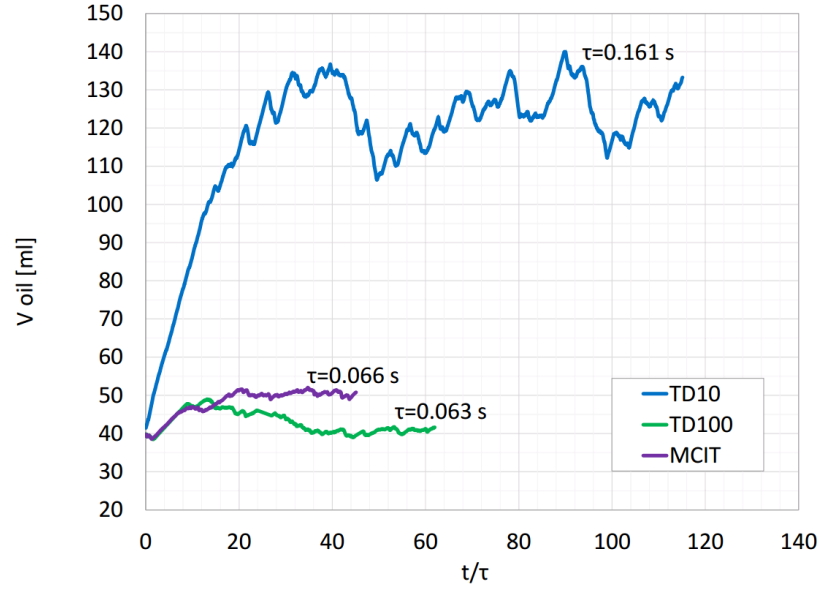


Figure 4.16 – Time evolution of oil residence volumes for different turbulence damping approaches

Table 4.5 – Computed liquid quantities

	TD100	TD10	MCIT
U_0 [m/s]	2.53	1.23	2.49
h_0 [m]	$0.25 \cdot 10^{-3}$	$0.72 \cdot 10^{-3}$	$0.37 \cdot 10^{-3}$
λ [-]	0.047	0.796	0.104
F [%]	2.3	7.6	2.9

shear can no longer maintain the unidirectional flow of oil on the wall taking place at lower λ values (see also Kakimpa et al. [51] for an extensive analysis of the shock solution).

A recirculation region starts to form, leading to oil accumulation. This behaviour has been observed in the simulation when switching from the TD100 to the TD10 case. Oil re-distributes starting from a thin film covering the front half of the chamber (Figure 4.17(a)) to a large, unsteady thick film region in its rising side (Figure 4.17(b)). The growth of the recirculation region explains the increase in oil volume observed in the TD10 case, as opposed to the TD100 and MCIT cases. The unsteady behaviour of this region justifies

also the larger oscillations in residence volume, as oil tends to build-up on the chamber wall before being ingested in the scavenge pipe. Despite the different turbulence damping approach, the oil film distribution and residence volume of the MCIT case (Figure 4.17(c)) are a close match to the TD100 case instead.

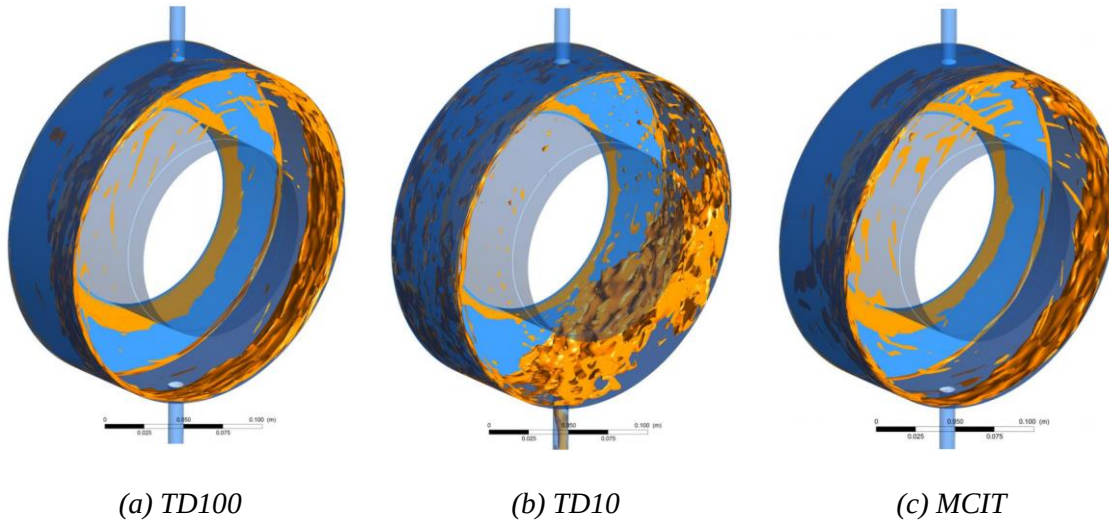


Figure 4.17 – Oil film distribution for different turbulence damping approaches . Contours of iso-surfaces of $\alpha_o = 0.5$: (a) $t/\tau = 44.7$; (b) $t/\tau = 138.3$; (c) $t/\tau = 37.7$.

Scavenge Efficiency

The scavenge efficiencies have been computed using Equation (4.3), considering the time-averaged scavenge mass flow rates. Table 4.6 summarizes the results for the three cases analysed, as well as the reference experimental data. The parameter $\Delta t/\tau$ represents the length of the time-averaging interval, selected for each case to match stationary oil residence volumes.

Table 4.6 – Effect of turbulence damping on scavenge efficiencies

	TD100	TD10	MCIT	Experiment [10]
$\Delta t/\tau$ [-]	2.52	5.59	4.54	-
η_{sc} [%]	10.3	90.5	15.6	37.3 ± 4.7

The results highlight a tendency of both TD100 and MCIT cases to under-predict the scavenge efficiency, while in case TD10 this is largely over-predicted. In the experimental observations [10], for the operating conditions presented here, oil tends to entrain with the sealing air flow and to leave the chamber mostly through the vent pipe, leading to a relatively low scavenge efficiency. This behaviour is reproduced in cases TD100 and

MCIT, although the oil venting estimation is excessive. Case TD10 shows most of the oil leaving the domain through the bottom of the chamber instead. The latter behaviour is consistent with a gravity-dominated oil flow, but further discrepant with the scavenge efficiency found experimentally.

Film Thickness Distribution

The film thickness averaging approach follows the procedure discussed in Section 4.2.7 and adopted in the baseline simulation (Section 4.4.2). Figure 4.18 reports the computed oil film thickness circumferential distributions for the cases analysed, compared against the reference experimental data [10]. The results confirm the distributions of Figure 4.17 and provide a quantitative match to them. The large accumulation zone of Figure 4.17(b) corresponds to the high oil film thickness peak in the region between 90° and 165° found on the TD10 curve in Figure 4.18. Cases TD100 and MCIT show a smooth film in the same area instead, thus confirming the shear-driven behaviour of low gravity parameter flows. A thickening of the film is observed at about 75° , but this is not associated with oil recirculation as found for TD10.

All three numerical methods tend to under-predict the oil film thickness in the co-current side, i.e. the region between 195° and 330° . Nonetheless, the experimentally observed thinning of the oil film as it gains speed towards the scavenge pipe is generally identifiable in the numerical simulations. Cases TD100 and MCIT better reflect this

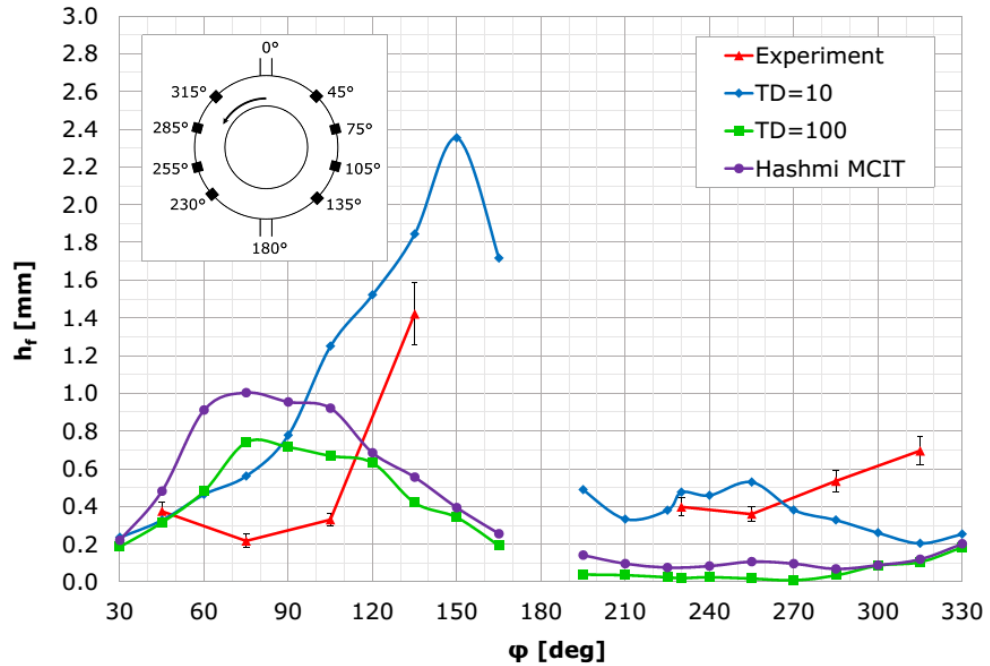


Figure 4.18 – Numerical and experimental oil film thickness distributions

behaviour, while TD10 shows a thickening of the film already at $\varphi = 315^\circ$. The trend in TD10 is due to the lower film speed, which results in a build-up of oil as this flows towards the scavenge pipe.

Considering the circumferential distributions, case TD10 has a better agreement with the experimental film thickness trends as opposed to cases TD100 and MCIT. The results suggest that the air shear forces in the experiment are not as dominant on the oil film behaviour as predicted by the latter two cases, thus gravity forces contribute significantly to generate oil accumulations in the counter-current side of the chamber. Nonetheless, the balance is now excessively shifted towards the latter forces, indicating that with Egorov's method [80] the ideal turbulence damping factor for this simulation should lie between $B = 10$ and $B = 100$.

All three cases show a non-uniform distribution of oil across the bearing chamber axial length. In particular, cases TD100 and MCIT have the bearing-side portion of the outer wall remaining dry (Figure 4.17 (a) and (c)). In real bearing chamber conditions, it is unlikely to have a uniform wall oil coverage in the axial direction because of the secondary flow generated by the sealing air [89]. However, as observed in Section 4.4.2, the lack of experimental data makes it challenging to assess the accuracy of the predictions. Thus, the numerical oil film distributions in the axial direction should be regarded cautiously.

Film Velocity

The velocities of the moving oil films have been computed at their interface with the air, namely the iso-surface of oil volume fraction $\alpha_o = 0.5$. The reference film speeds U_0 of Table 4.5 summarise the global film surface velocities for the three cases. It is observed that case TD10 has a global film velocity value which is about half that of TD100 and MCIT, reflecting its significantly lower gas velocity also on the oil film surface behaviour.

The contour plots of Figure 4.19 give a further insight in the spatial distribution of the time-averaged film surface velocities. Consistently with the relatively low reference film speeds, case TD10 has peak film velocities of the order of 1 m/s (Figure 4.19(b)) while values up to 3 m/s are found in cases TD100 and MCIT (Figure 4.19(a) and (c) respectively). The low film velocity of case TD10 is interlinked with the observation of recirculating behaviour of oil in the region next to the scavenge pipe. Thus, the film has highly unsteady characteristics, with portions of oil flowing back towards the scavenge pipe. This recirculating behaviour of case TD10 is highlighted in the plot of Figure 4.20,

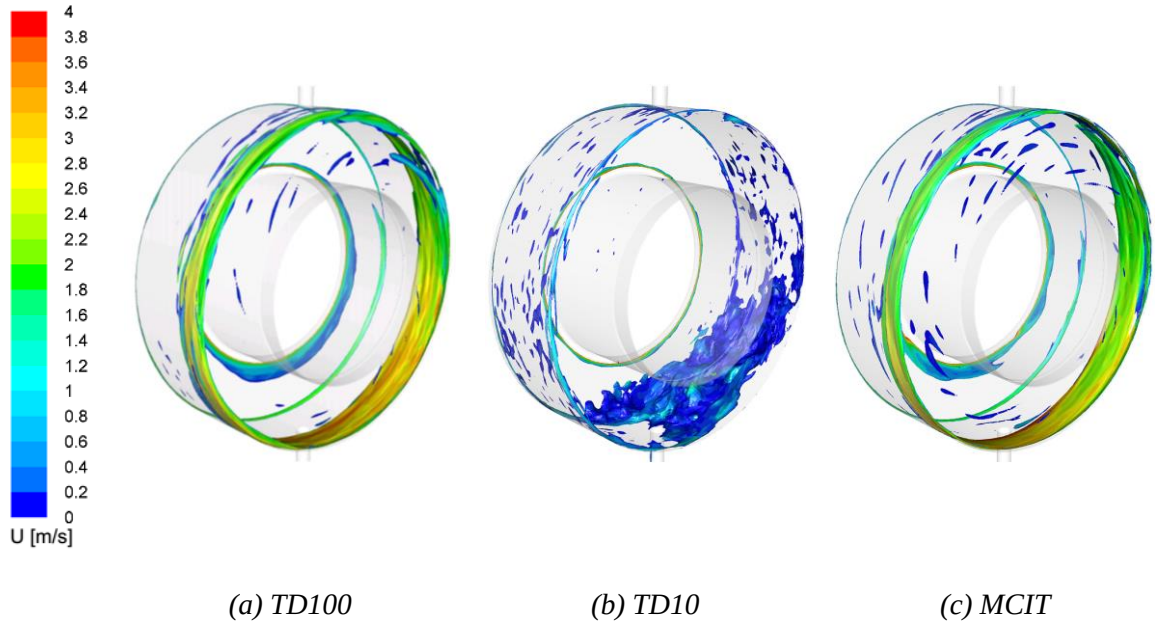


Figure 4.19 – Contours of oil film surface velocities at $\alpha_o = 0.5$, time-averaged over 0.1 seconds

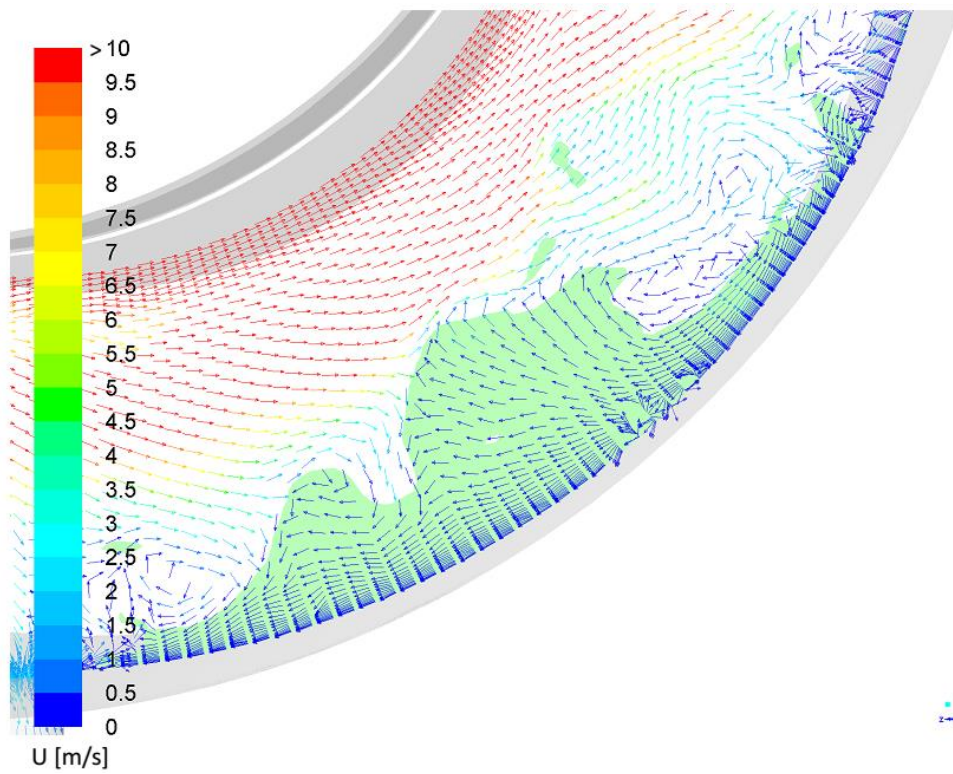


Figure 4.20 – Velocity vectors and oil distribution in cross-section of recirculation zone ,
Case TD10, $t/\tau = 111.5$

which shows the in-plane velocity vectors in the mid-section of the bearing chamber. Oil flows back to the scavenge pipe at relatively low velocity, i.e. less than 0.5 m/s, as the air flow cannot provide sufficient momentum for the film to reach the top part of the chamber. The result is a region where oil is nearly static and tends to build-up.

The different oil behaviour of case TD10 over cases TD100 and MCIT highlights that the turbulence damping approach plays a key role on determining the final oil film velocity and, in turn, whether the predicted flow behaves as gravity- or shear-dominated. As the two different outcomes have a significant impact on the accuracy of the bearing chamber predictions, more focus is needed on this aspect of VOF for it to be readily deployed for design.

Effect of turbulence damping

Since the turbulence damping approaches evaluated in this work control the momentum transfer at the free surface by either directly or indirectly modifying the turbulent viscosity, the latter parameter has been considered to better understand the effect on the flow field. Turbulent kinetic energy has been also evaluated, as it provides information on the average velocity oscillations in the domain.

Table 4.7 – Turbulent quantities computed in bearing chamber annulus

		TD100	TD10	MCIT
$\overline{\mu_t}$	$\left[kg \frac{m}{s} \right]$	$2.4 \cdot 10^{-3}$	$4.0 \cdot 10^{-3}$	$2.1 \cdot 10^{-3}$
$\int_{V_{ann}} TKE \rho dV$	$\left[kg \frac{m^2}{s^2} \right]$	$3.2 \cdot 10^{-2}$	$3.1 \cdot 10^{-2}$	$1.7 \cdot 10^{-2}$

The comparison of the turbulent quantities summarised in Table 4.7 demonstrates that the turbulence damping value introduced in case TD10 increases the average turbulent viscosity of the fluid when compared to TD100, giving a near-doubled $\overline{\mu_t}$ value. This result is in-line with the expectations, as a low turbulence damping factor B in Equation (3.86) gives a smaller correction of the specific dissipation ω . The consequence is an increase of the interface μ_t , which eventually affects the whole bearing chamber turbulent viscosity. One direct effect of the relatively high average turbulent viscosity value of case TD10 is the low average gas velocity found in the bearing chamber annulus, which explains the relatively high value of τ reported in Figure 4.16.

Whilst cases TD100 and MCIT lead to near-matching $\overline{\mu_t}$ values, the mass-weighted integral of the turbulent kinetic energy (TKE) does not show an analogous match. The MCIT approach yields a lower TKE integral in the bearing chamber annulus, meaning that the velocity oscillations due to turbulence are lower in magnitude. Despite the similar outcome in terms of oil film distribution and flow velocities, cases TD100 and MCIT

therefore lead to flow predictions which differ in terms of turbulent kinetic energy content. The reason for this difference is that while Egorov's turbulence damping approach [80] modifies only the ω equation of the turbulence model, leaving the turbulent kinetic energy unaltered, Hashmi's MCIT [39] changes both k and the turbulent dissipation (ϵ in our case) to obtain a similar wall-like behaviour at the interface (see Section 3.5.3).

Conclusions

Considering Egorov's approach [80], the results highlight the tendency of the liquid flow to switch from an air shear-driven to a gravity-driven condition as the damping factor is decreased. This suggests that the physics of the problem are imposed, rather than predicted, when selecting the turbulence damping factor value. The results obtained using the MCIT approach show that, despite its complex formulation, this method does not bring a significant improvement in accuracy of the results compared to Egorov's [80]. As experimental data for comparison of the turbulent quantities in a bearing chamber is not available in the literature, it is still difficult to determine which of the three cases reproduces more accurately the real bearing chamber flow conditions.

Since Egorov's damping method [80] provides at least an input factor to tune its effects, the information gathered in the test cases recommended to use the latter with a value of B lower than 100 for the successive simulations. For this reason, the BR725 bearing chamber presented in Chapter 5 relies on a turbulence damping factor value of 50, selected as intermediate between the two values tested. As this approach is purely empirical, a future, more robust solution would be to match the predictions of the VOF method with those of a model for which the computation of the gas-liquid interface quantities is known to be accurate. An example is the one-dimensional thin film model of Kakimpa et al. [51]. This would also enable to extend VOF modelling to geometries that are different from those for which experimental data is currently available.

An experimental set-up to reproduce air-driven rimming flows in a closed environment, i.e. without any inlet or outlet, would be desirable to generate additional data for validation. This set-up would inherently exclude flow disturbances such as oil and air inlet/outlet flows and unsteady oil residence volumes, and would lead to a simpler numerical model characterised by shorter computational times.

This study on the effect of turbulence damping opened a series of questions on the suitability of the current guidelines for bearing chamber modelling [43]. Additional research to extend the validation of the VOF model for bearing chamber flow conditions

is recommended. This should include a wider range of tests using Egorov's approach [80], as well as Hashmi's Simplified Interface Treatment [39] after overcoming the stability limitations encountered in the preliminary tests of this work (see Section 3.5.3).

4.5 Heat Transfer

In order to assess the heat transfer modelling on bearing chamber flow conditions, a simulation has been set-up starting from the solution of the baseline case in Section 4.4.2. To set the boundary conditions, information has been shared by Rolls-Royce Deutschland and Dr Wolfram Kurz, who provided more details about the experimental set-up of the results published in [10] in a personal communication in 2015 [38]. The latter experiment included a heat input of about 100 W on the bearing chamber outer and air inlet-side walls, to keep the oil temperature constant at about 100°C. The heat transfer simulation has been set up with the same heat power input to evaluate qualitatively the wall temperature and heat transfer coefficients distribution, as well as their correlation with the corresponding oil film thicknesses.

In addition to the settings reported in Section 4.4.2, a heat input per unit area of 1394 W/m² is added on the chamber outer and air inlet-side walls to obtain the total input of 100 W. As the heat propagation in the chamber walls is not an objective of this test, the wall heat conduction modelling has not been implemented. The air and oil inlet temperatures are set to 373 K, while the oil outlet temperature at the bottom of the chamber is set 2 K higher than the inlet. Variable piecewise-linear oil properties within the range of 40°C–150°C had been originally selected, but stability issues led to switch them back to their 100°C fixed values for this test. To assess the achievement of the flow stationary state, stable vent and scavenge pipe outlet flow temperatures have been monitored.

After an initialisation with constant fluid temperature of 373 K in the whole domain, the simulation required about 0.2 seconds of flow time to reach stable vent and scavenge outlet temperatures. The plot of Figure 4.21 reports the time evolution of the two temperatures in the control surfaces considered. The location of the control surfaces is also shown in the figure. The trends of the outlet fluid temperatures indicate that the flow in the scavenge pipe tends to overheat before settling to the final temperature. The reason for this initial behaviour appears connected with the numerical instabilities found in the first iterations of the simulation, which also prompted to reduce the time step size by a

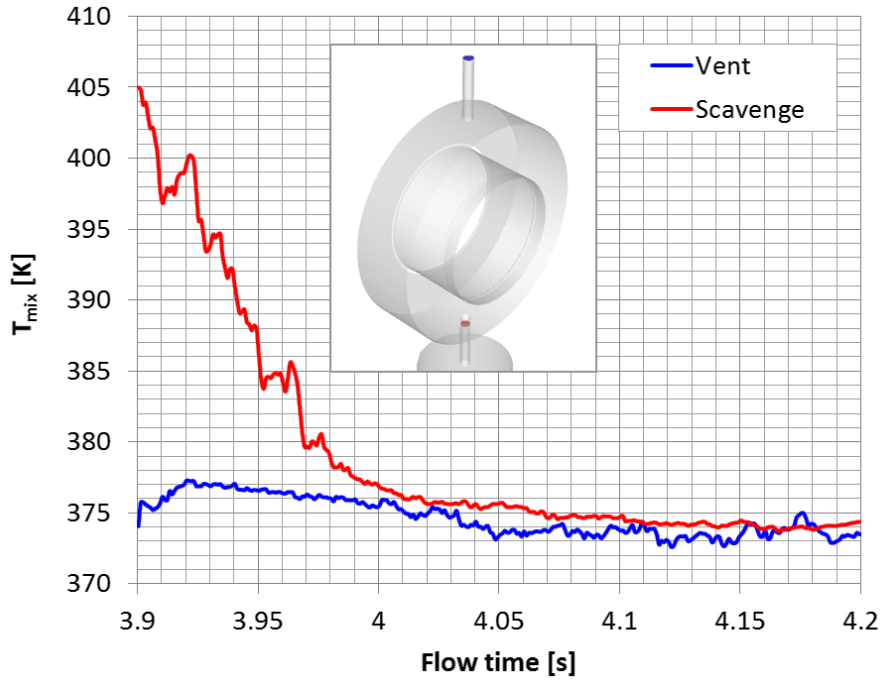


Figure 4.21 – Time-evolution of bearing chamber outlet temperatures in vent and scavenge control surfaces

factor of 100 to achieve convergence in the early stages. Consistently with the expected small increase relative to the inlet temperature, the final outlet temperatures stabilise at about 375 K in both pipes.

Figures 4.22(a) and (b) report respectively the heat transfer coefficients (HTCs) and the static temperature distribution for the two heated walls of the bearing chamber. Considering the chamber outer wall in Figure 4.22(a), two clear ‘strips’ with high HTCs are found on the two outer sides of the wall. These regions correspond to the areas of high oil coverage shown in Figure 4.22(c). Areas where the coverage is lower, i.e. where the oil films are thinner than the near-wall cell height, have lower HTC values. From a physical point of view, this prediction is in line with the expectations, as thin films tend to be laminar and therefore provide a lower turbulent heat transfer capacity as opposed to thick films. In this simulation, the difference between heat transfer coefficients in thick and thin films is approximately of the order of $3000 \text{ W}/(\text{m}^2 \text{ K})$, as the thick film regions yield HTCs of about $8000 \text{ W}/(\text{m}^2 \text{ K})$, against the average $5000 \text{ W}/(\text{m}^2 \text{ K})$ found in the thin films.

Figure 4.22(b) reports the wall static temperature distributions. As opposed to what the heat transfer coefficients contours would suggest, the areas with the lowest HTC values correspond to those where the wall temperatures are lower. The reason for this behaviour is found in the fluid temperature adjacent to the wall: while the thick film-covered regions

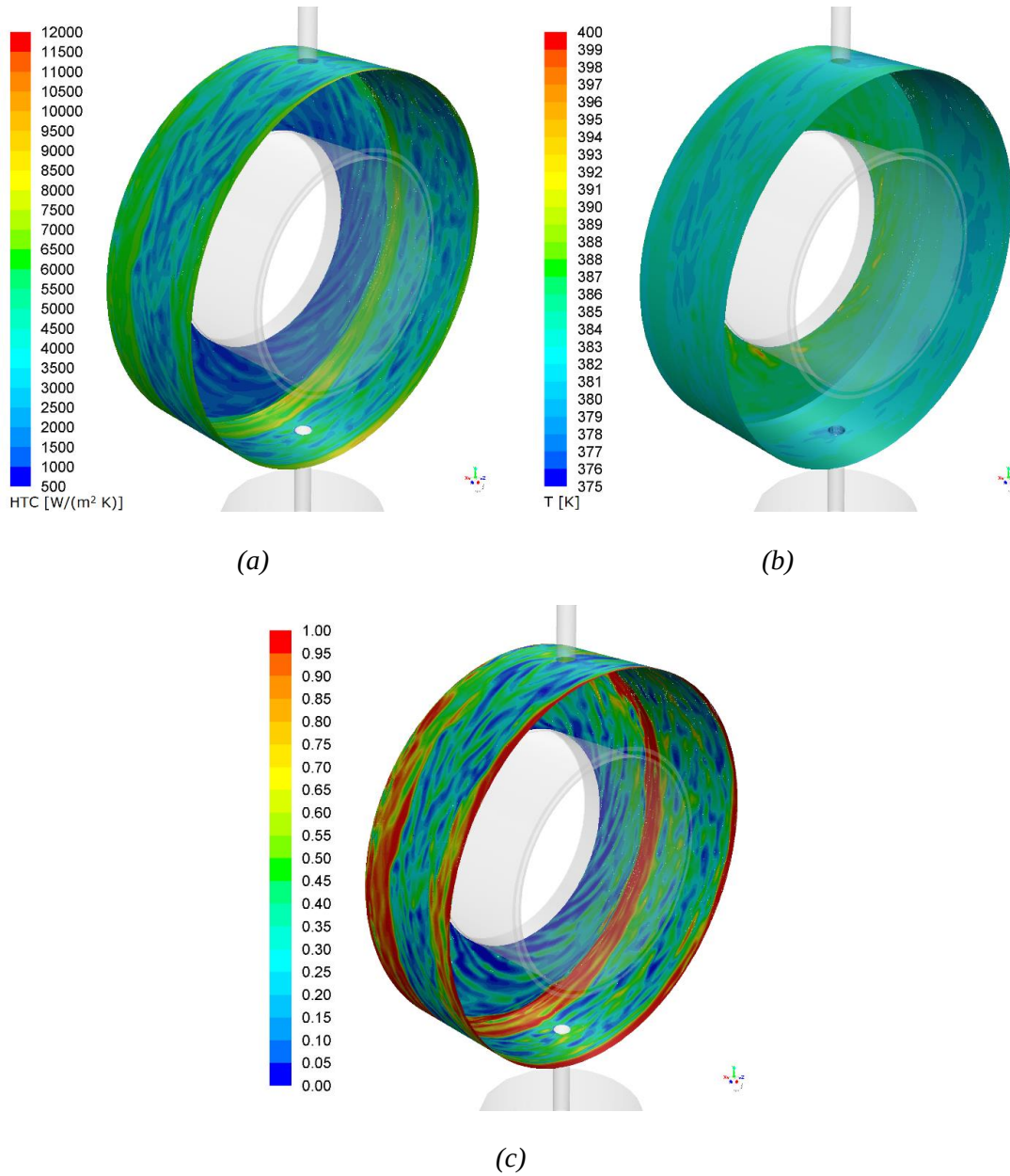


Figure 4.22 – Bearing chamber thermal simulation results, contour plots on heated walls;
 (a) Heat transfer coefficients, (b) Static temperatures, (c) Oil volume fractions

are characterised by a layer of oil that tends to heat up as it does not leave the chamber, the thin film regions are in contact with a fast air flow which is rapidly expelled. From the one-dimensional Fourier's law of convective heat transfer,

$$q = HTC \cdot A \cdot \Delta T \quad (4.6)$$

having fixed the wall heat flux q in the simulation, lower HTC values imply that the wall-fluid temperature difference ΔT needs to be higher. This would normally mean that the wall starts to overheat, but the contact with the air flow in the thin film regions provides a sufficient temperature gradient to ensure the required heat flux.

From the observations above, it can be stated that while thick oil films provide higher wall heat transfer coefficients, their enhanced cooling effect is beneficial only if an appropriate exchange of oil, i.e. avoiding recirculation, is provided. The sealing air flow is found to have a strong effect in the bearing chamber cooling, because of the rapid exchange of air in the annulus which favours the extraction of heat. The latter statement is aligned with the observations made by Glahn et al. [13], who found that an increase of the sealing air flow has a stronger cooling effect than an increase in the oil flow rate. The computed heat transfer coefficients are of the same order of magnitude as the values found in the experiment of Busam et al. [12], which are in the range of 2000–4000 W/(m² K).

The results confirm that realistic wall heat transfer coefficients and temperature distributions can be obtained with the approach followed. The outcome is in line with the observations found in previous experimental studies [12, 13] and in agreement with the flow conditions found in the bearing chamber. Given the successful testing on the KIT bearing chamber model, the heat transfer modelling procedure defined in this section has been implemented in the BR725 bearing chamber simulations of Chapter 5. For stability reasons, constant wall temperatures instead of heat fluxes have been used in the latter case.

4.6 Summary and Discussion of the Results

The set of simulations presented in this chapter assessed the performance of the VOF method on the analysis of an experimental bearing chamber at high shaft speed. The study showed that the implicit formulation of the method is the only suitable way to simulate transients of the order of several seconds on a sufficiently fine mesh. The Compressive interface reconstruction method has been used in combination with this formulation, proving suitable to keep a sharp gas-liquid interface and avoiding numerical diffusion. Heat transfer modelling has been implemented, showing good agreement with the data available in the literature [12, 13].

The results highlighted that the choice of the turbulence damping method plays a fundamental role in defining the characteristics of the oil flows inside the bearing chamber. It has been demonstrated that a simple variation of turbulence damping factor as an input using Egorov's method [80] can make the oil flow switch from shear-driven to gravity-driven. The uncertainty linked with the turbulence damping selection can have a strong impact on the accuracy of the results, and suggests that there is a risk of imposing the oil flow conditions rather than predicting them. If there is no sufficient information about the

expected regime for a given bearing chamber, this choice may lead to erroneous predictions of the oil distribution and bearing chamber scavenge efficiency. Other methods such as Hashmi's MCIT [39] did not show particular benefits in these terms, adding the uncertainty of whether a model without any factor to control the level of gas-liquid turbulent interaction is sufficiently mature to capture the complexity of a bearing chamber flow.

The work in this chapter showed also that established methodologies to select the turbulence damping factor, based on the work of Tkaczyk [43], cannot be directly transposed from a stratified channel flow to a bearing chamber flow while expecting the same level of accuracy. This suggests that further investigations should be based on shear-driven liquid flows that are better representative of bearing chamber flow conditions, such as rimming flows.

For the reasons above, the level of understanding and maturity of the turbulence damping approach when implemented onto bearing chambers does not ensure yet a fully robust application at industrial level. Additional work on this aspect is needed to achieve the required technology readiness level (TRL) 6. This should include a dedicated experiment for CFD validation, as will be discussed in Section 6.3. This could be implemented relatively easily on an existing test rig where a stub shaft is available, for example the G2TRC bearing chamber rig [24-30] or the one introduced by Ahumada et al. [130].

For the purpose of this work, the evaluation of the VOF method on the real bearing chamber of Chapter 5 has been carried out using Egorov's turbulence damping method [80]. Based on the results obtained in the KIT bearing chamber simulation, and considering the dimensional similarities, the selected damping factor has been identified among the values that should best reproduce the expected flow regime.

4.7 Simulation Guidelines

Introduction

This section is intended to provide a quick reference for the set-up of a bearing chamber simulation using the VOF method. Some of the contents of the previous sections are recalled here in order to provide a complete guide that can be used independently from the contents of the whole chapter.

Context Diagram

The analysis of all the different factors affecting the bearing chamber film thickness distribution, the air flows, and the simulation time requirements, has led to the creation of a context diagram to summarise the parametric studies performed on the KIT bearing chamber. The diagram, shown in Figure 4.23, reports the current understanding of the factors influencing the system as well as their weight. It includes also the lessons learnt from short test cases that, for practical reasons, have not been included in the previous sections of this chapter.

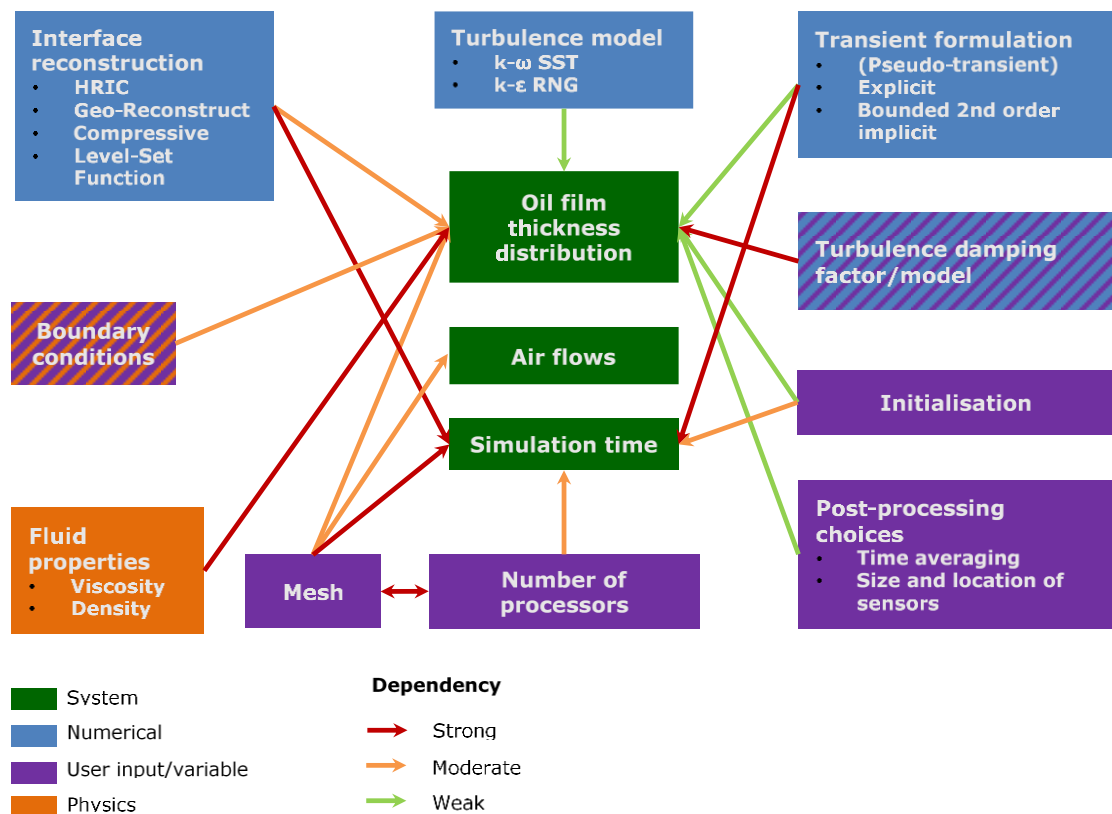


Figure 4.23 – KIT bearing chamber context diagram

Model Choice

The context diagram highlights that, to the current understanding, one of the major user-defined choices affecting the oil film thickness distribution is the turbulence damping approach and, if required, the selection of its scaling parameter. For the time being, based on the results of Section 4.4.4, Egorov's turbulence damping [80] is the recommended approach for the modelling of bearing chambers having size and operating conditions comparable with the KIT chamber. The proposed damping factor value is $B = 50$. Because of the dependence of this factor on the mesh size and flow conditions, however,

preliminary sensitivity studies are recommended before starting a new application on other components of the oil system.

Considering the speed-up of the simulation times, the implicit VOF formulation, with Compressive interface reconstruction and bounded second order implicit time formulation, gives rapid convergence with time step sizes up to five times larger than those required by the conventionally-adopted explicit VOF with Geometric reconstruction. The reconstruction accuracy of the Compressive scheme has been evaluated in Section 4.4.1, as well as demonstrated in relevant applications available in the literature [9, 34, 71].

Mesh

According to the recommendations of Tkaczyk [42], a minimum of six cells should cover the liquid film thickness to accurately describe its velocity profile. The tests performed in Section 4.4.3 have shown that further refinements do not bring any additional improvement in the accuracy of the results. Therefore, it is recommended to design the near-wall mesh following this indication, consistently with the desired minimum film thickness to be captured. For bearing chamber flow conditions, the near-wall cell height should be in the range of 50 to 100 μm to meet this requirement. The design of the mesh should take into account, however, the desired wall y^+ values, which should be around 1 if heat transfer modelling is required.

In the regions of the annulus which are far from the walls, the study has shown that for a dry chamber the cell sizes should be of the order of 2 millimetres to achieve mesh independence for the secondary air flows (see Section 4.3). When implementing this guideline in a two-phase flow, however, it should be kept in mind that small vortices generated by the interaction of air with oil films may require finer meshes to be accurately described.

Time step

As anticipated in the ‘Model Choice’ section, the implicit VOF formulation enables higher Courant number values in the domain as opposed to the explicit formulation. While the latter starts to have convergence issues for $\text{CFL} > 5$ (see also Krug [92]), the implicit formulation can be used for CFL values up to 20. It should be reminded, however, that the fluid regions with Courant number higher than 1 should be of limited size and cover only locations where the solution does not require high accuracy. In the cases simulated, high Courant numbers are reached only at the vent pipe inlet, where the results are believed to have only a small influence on the oil film distributions.

With exception for the start-up of the simulation, where small time steps can be used, acceptable time step sizes are of the order of 10^{-5} seconds or higher. Smaller time steps are likely to lead to excessively long computational times before achieving the flow stationary state, which normally requires multiple seconds of flow from the beginning of the simulation.

Initialisation

Different initialisation approaches, e.g. starting from a patched oil film or from a previous solution, have shown no particular dependence of the solution from the initial status, provided that sufficient time is allowed for the flow to stabilise. A more realistic flow initialisation consists in letting the oil flow ‘drip’ from the bearings (i.e. without shaft rotation and sealing air flow) until the residence volume in the chamber is stable. This approach has been suggested by Peduto [41] and has been successfully implemented on the BR725 bearing chamber, as reported in the following chapter.

Data Processing

In order to get a detailed insight of the oil film thickness distribution, its evaluation at a number of discrete circumferential locations which is higher than those reported in the reference experiment [10] is recommended. Studies to evaluate the sensitivity of the results to the circumferential and axial positions of the film thickness sampling areas are desirable, as oil may tend to accumulate in certain regions of the domain that are not necessarily covered.

The film thickness distributions should be time-averaged over a sufficiently long time interval (of the order of 0.5 seconds) to ensure that time-dependent variations at the given sensors’ positions are averaged. Using a manual time-averaging method or the ‘Data sampling’ option available in ANSYS Fluent has given equivalent results (see Section 4.2.7). As the latter option averages the results within the solver, it is the recommended choice as it avoids the production of hundreds of data files containing the iso-surfaces of the volume fractions, which may rapidly generate storage problems.

Considering the sampling of the oil flows at different control planes in the outlet pipes, it has been found that using multiple planes does not generate different results than relying on a single one. In order to account for all the time steps while retaining relatively small output files, the time-averaging option in the output monitors of ANSYS Fluent is recommended for both the outlet flows and for the oil residence volume.

5 Application: Rolls-Royce BR725 Rear Bearing Chamber

5.1 Introduction

This chapter covers the application of the VOF approach presented in Chapter 4 onto the Rolls-Royce BR725 Rear Bearing Chamber (RBC). This constitutes a demonstration exercise aimed at evaluating the performance of the model, the challenges of setting-up a complex bearing chamber configuration, and the unresolved issues needing specific attention for its successful deployment.

The Rolls-Royce BR725 is a two-shaft engine designed to power long-range, large business jets. The oil system design is particularly important for this type of engines as, on top of increasing oil consumption, venting overboard large amounts of oil is highly unwelcome by the customers of this market segment. Bearing chambers play a significant role in separating oil from the air/oil mixture that is vented overboard, as they perform the first stage of separation upstream of the deaerator and breather (see Section 1.2). This study focuses on the rear bearing chamber of the engine, where two sets of roller bearings hold the LP and HP shafts in place. This chamber has been selected because of the particularly harsh operating conditions it faces, as it is located in one of the hottest regions of the engine core. A better understanding of the flow conditions taking place can be then used to inform the design of future bearing chambers. It is of particular interest to visualise the oil flow behaviour around the vent pipe. The latter has a peculiar arrangement as, unlike the Trent-type bearing chambers, it is protruding from the bottom of the annulus. An additional advantage of selecting this chamber is that the results can be compared with the ones obtained on the KIT bearing chamber in Chapter 4. In fact, the two bearing chambers are characterised by size and operational similarities.

5.2 Objectives

The objectives of the work in this chapter are the simulation of the air-oil flows for two bearing chamber operating conditions, namely ground idle and maximum take-off power (MTO), as well as the implementation of the energy equation (see Section 3.2.4) to better understand the heat transfer mechanisms taking place in the chamber. The results

are validated against the design assumptions and, where possible, against the measurements made in the engine validation tests.

5.3 Set-up

5.3.1 Bearing Chamber Geometry

The BR725 Rear Bearing Chamber (RBC) is characterised by two shafts held in place by two sets of roller bearings. The bearings are lubricated by one oil jet per set, with the flow coming from a so-called jet block located at the bottom of the chamber, as well as from the bearing squeeze film dampers. Next to the jet block is located the scavenge port, of the covered ramp type, and the protruding vent pipe. Figure 5.1(a) illustrates the arrangement of these three components.

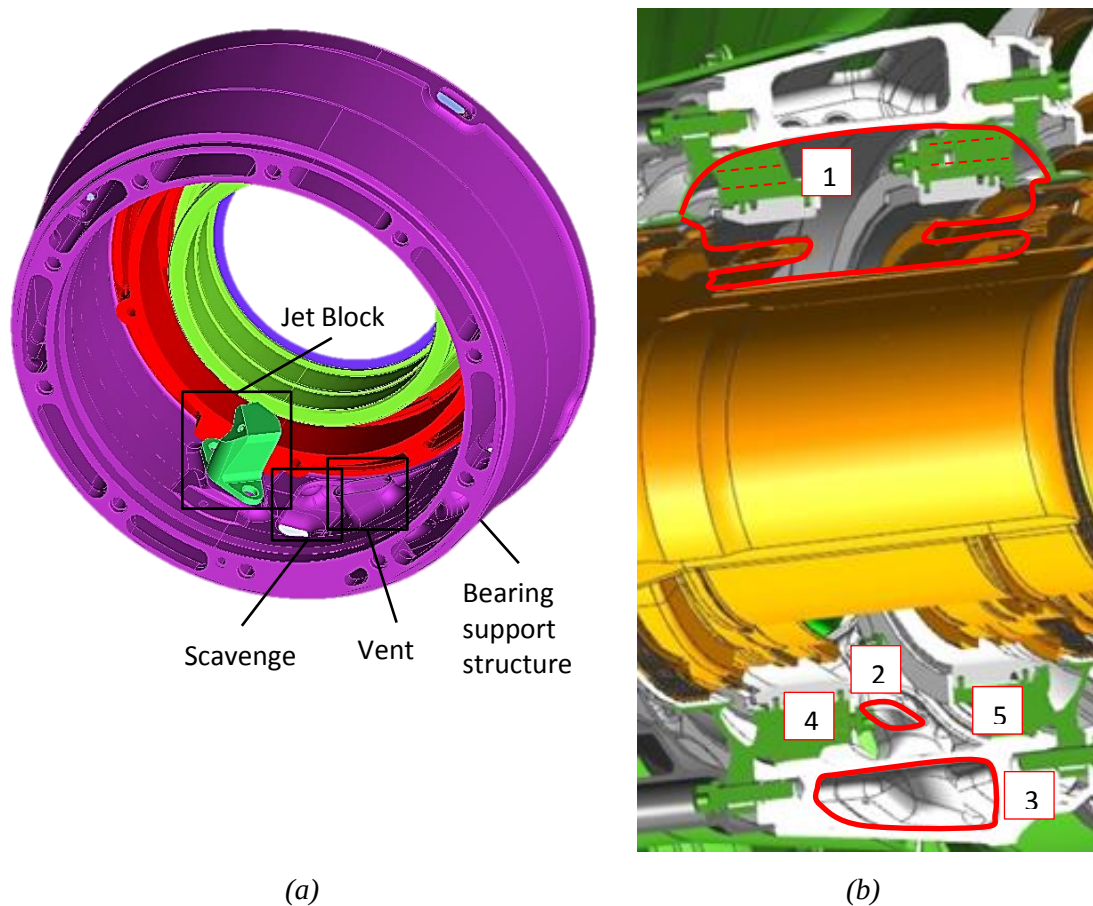


Figure 5.1 – CAD model of Rolls-Royce BR725 Rear Bearing Chamber. (a) Bearing chamber structure without shafts; (b) Cross-section of the chamber including shafts – 1: bearing support structure, 2: vent pipe inlet, 3: scavenge cavity, 4-5: LP and HP bearing outer races

The chamber annulus has a cross-section which resembles a horizontal H-shape, as illustrated in the top of Figure 5.1(b). Radial struts are located all around the annulus to

support the bearings. The presence of these structures creates axial slots inside the chamber through which the air/oil mixture can flow. The slots are part of the bearing support structure shown in Figure 5.1(a); their direction is represented by the dashed lines in Figure 5.1(b). Table 5.1 summarises the bearing chamber relevant dimensions.

Table 5.1 – Rear Bearing Chamber relevant dimensions

Chamber axial length	b	129.0	mm
LP shaft length	b_{LP}	116.4	mm
HP shaft length	b_{HP}	40.9	mm
Outer wall max diameter	$\varnothing_{rw,max}$	198.0	mm
LP shaft max diameter	$\varnothing_{LP,max}$	150.4	mm
LP shaft min diameter	$\varnothing_{LP,min}$	107.2	mm
HP shaft max diameter	$\varnothing_{HP,max}$	150.4	mm
HP shaft min diameter	$\varnothing_{HP,min}$	120.0	mm
Vent pipe – cross-sectional area	A_{vt}	187.25	mm ²
Scavenge pipe – cross-sectional area	A_{sc}	188.16	mm ²
Scavenge outlet – cross-sectional area	$A_{sc,out}$	7848	mm ²
HP shaft seal – air inlet channel thickness*	$h_{a,HP}$	0.375	mm
LP shaft seal – air inlet channel thickness*	$h_{a,LP}$	0.375	mm
HP-LP shaft seal – air inlet channel thickness*	$h_{a,HP-LP}$	1.835	mm
Oil inlet channels thickness*	$h_{o,in}$	3.000	mm

* In computational domain

The geometry of the fluid region, extracted from the CAD model, has been provided by Rolls-Royce Deutschland. Figure 5.2 shows the main features of the CFD model. Compared to the real engine geometry, the following simplifications have been introduced:

- Oil is injected as a continuous film from the bearings;
- The bearings are not explicitly modelled;
- The labyrinth seals are not explicitly modelled;
- The heads of the internal bolts have been removed.

With exception of the simplifications above, all the geometrical features of the real bearing chamber have been retained in the CFD model.

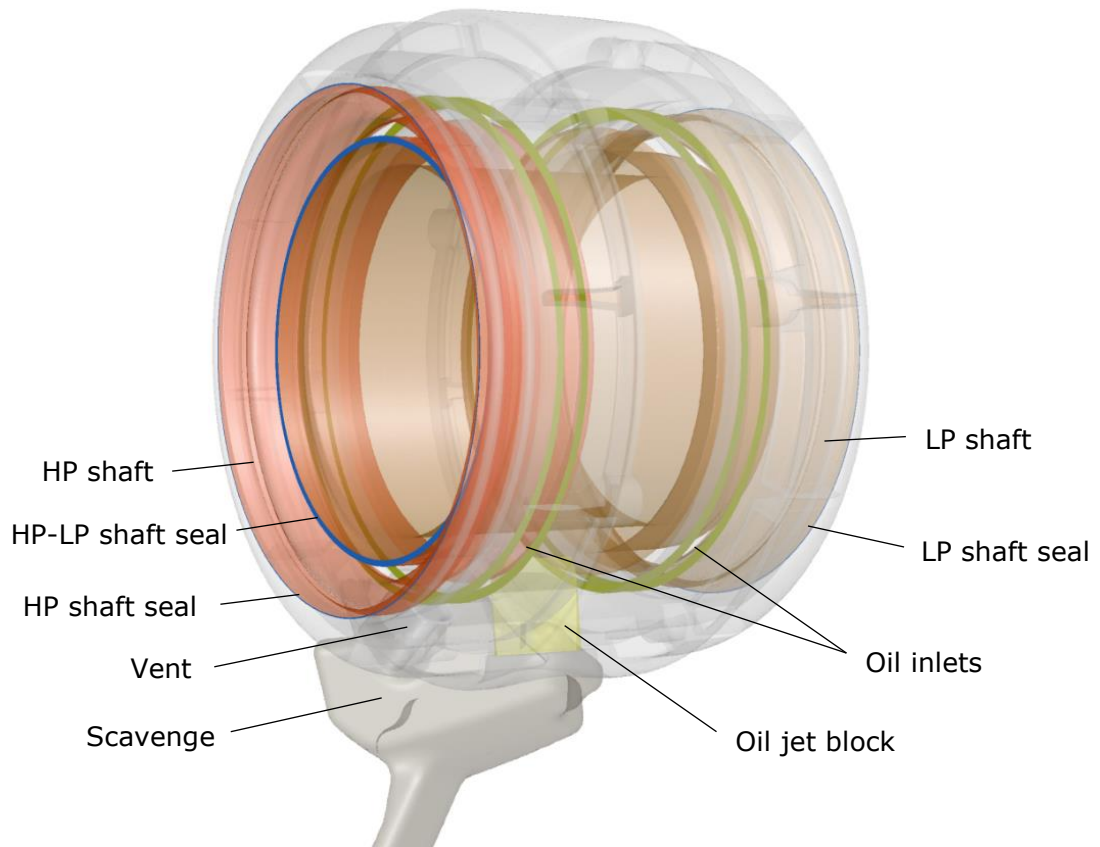


Figure 5.2 – CFD model of BR725 Rear Bearing Chamber

5.3.2 Mesh

Given the size and complexity of the bearing chamber geometry, it was decided to take advantage of the new meshing features available in ANSYS Fluent (starting from version 14.5) to create an unstructured, polyhedral mesh for the model, building on the experience of Piazero de Carvalho [131] in the Group. This type of mesh brings the major benefit of reducing the cell count required to achieve the same solution accuracy of an equivalent hexahedral or tetrahedral grid [132]. The mesh is also better suited to the calculation of recirculating flows, thanks to the increase of optimal directions through which the flow can enter a cell. Compared to a hexahedral mesh, it gives a more accurate computation of the fluxes into the control volume due to the high number of faces. This enhances the accuracy of the computation of gradients, which in turn improves the convergence properties of the model [132].

To generate the mesh, the following procedure has been observed. After cleaning and preparing the geometry provided with the help of CADFix [133], a dedicated tool available at Rolls-Royce, a first surface mesh with triangular elements has been created with ANSYS

ICEM [134] to capture the geometrical features of the model. The grid has been then imported into the meshing module of ANSYS Fluent, where a surface re-mesh has been performed before generating a tetrahedral volume mesh with near-wall prism layers. The grid has been successively converted into polyhedral mesh using ANSYS Fluent (solver), reducing the cell count by about 47% (18.5 to 9.9 million) while improving the element quality (minimum orthogonal quality increased from 0.01 to 0.13, maximum orthogonal skewness decreased from 0.97 to 0.87, see Table 5.2).

Prism layers have been created in the regions of interest of the bearing chamber. These include the chamber outer wall, where oil is expected to accumulate and form films, and the two shafts, where it was decided to have an accurate control of the wall y^+ to keep it into the $1 < y^+ < 5$ range. Analogously to the strategy adopted for the KIT bearing chamber in Section 4.3, preliminary studies have been performed to define the final layout adopted for all the two-phase and heat transfer simulations. These studies are presented in Section 5.4. Table 5.2 reports the main features of the mesh.

Table 5.2 – Mesh characteristics

Mesh size	$9.87 \cdot 10^6$	cells
Maximum cell volume in annulus	4	mm^3
Minimum orthogonal quality	0.13	[-]
Maximum orthogonal skewness	0.87	[-]
Maximum aspect ratio	110	[-]
Outer wall: Number of prism layers	8	layers
First layer height	~ 0.1	mm
Jet block: Number of prism layers	8	layers
First layer height	~ 0.1	mm
LP shaft: Number of prism layers	5	layers
First layer height	0.1	mm
HP shaft: Number of prism layers	5	layers
First layer height	0.05	mm

Figure 5.3 shows an overview of the polyhedral surface mesh; a cross-section of the volume mesh is provided in Figure 5.4. In the latter, the two axial extremities of the chamber outer wall are free from prism layers, as the software could not generate prisms

in proximity of the air inlet regions because of the tight spacing involved. For the same reason, the cell height on the prism layers of the outer wall and jet block is aspect ratio-based, which means that a fixed aspect ratio is used to size the cells in the layer. The result is that the layers tend to “inflate” where the surface mesh is coarser, which brings the additional advantage of having thick layers where most of the oil accumulation is expected, i.e. in the mid-section of the chamber. This meshing approach proved more robust where complex geometrical features such as small curvatures and slots are present. On the shafts instead, a fixed cell height could be maintained when generating the prism layers.

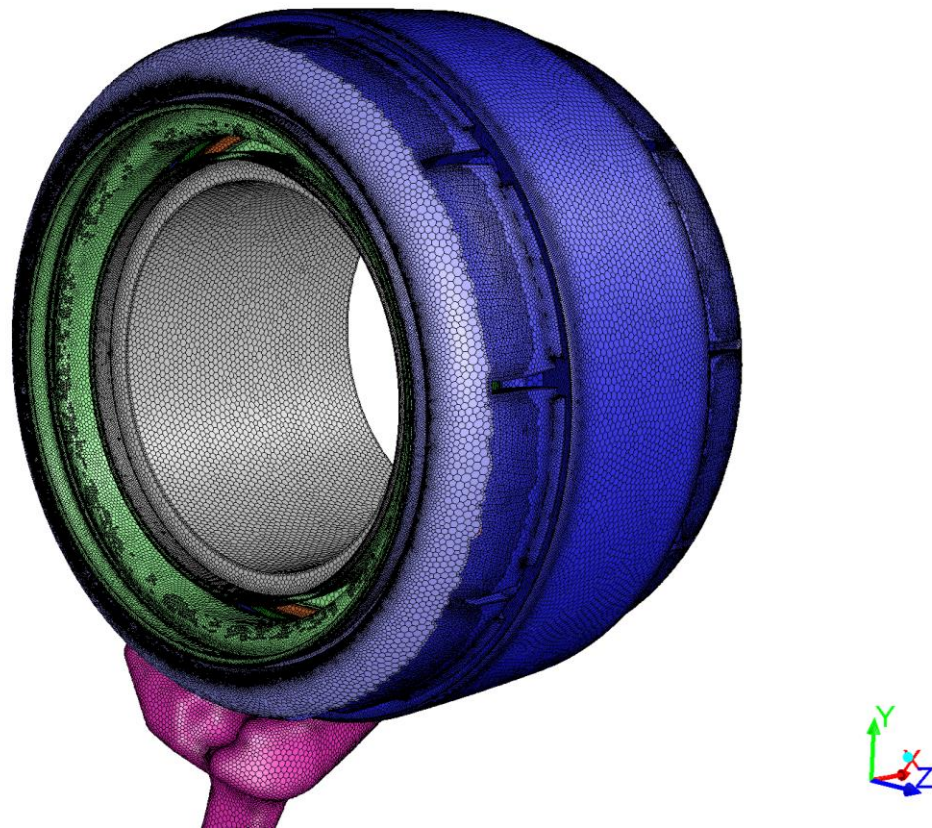
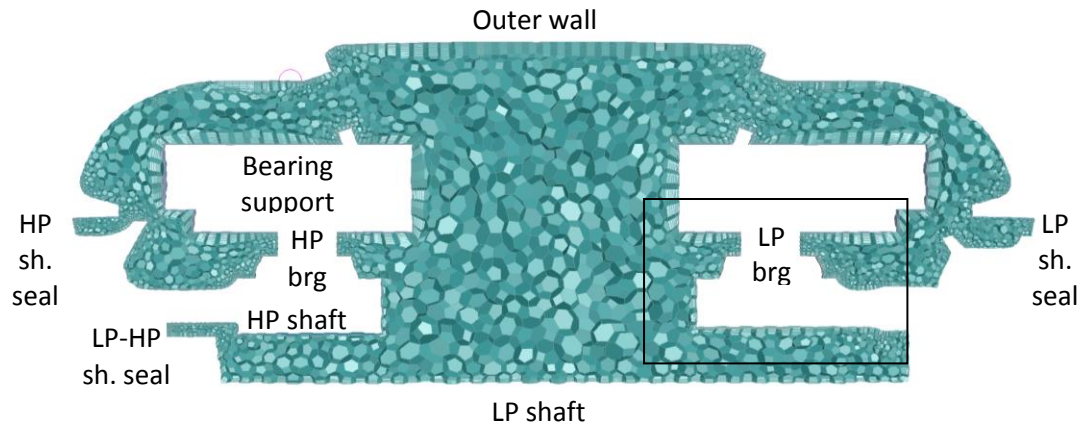
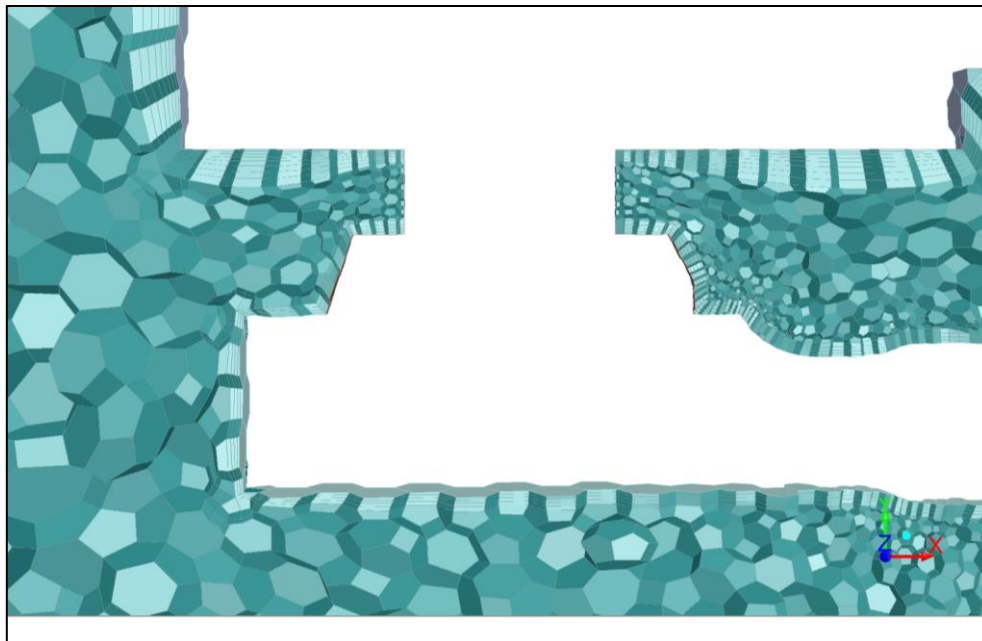


Figure 5.3 – Bearing chamber polyhedral mesh



(a)



(b)

Figure 5.4 – Cross-section of volume mesh in *xy* plane, top part of chamber annulus. (a) Overview of full section; (b) LP shaft bearing, magnified from (a)

5.3.3 Operating Conditions

To evaluate the bearing chamber behaviour in different flow and thermal regimes, two engine power settings representative of the range of operating conditions encountered by the chamber have been simulated. The first condition is ground idle, while the second is maximum take-off (MTO). The parameters needed for the set-up of the fluid dynamics simulations have been obtained from the engine design specifications and validation tests. The thermal simulation set-up relies on data from the internal Rolls-Royce thermo-fluids model SC03 for the chamber wall and oil inlet temperatures, while the air inlet temperature

has been obtained from the engine validation tests. The choice of the parameter values is the result of multiple interactions and discussions with the industrial supervisors.

Table 5.3 reports respectively the chosen fluid dynamics and thermal operating conditions of the bearing chamber, for both power settings. Table 5.4 lists the physical properties of the fluids. As reported in the latter, in the thermal simulation some of the air properties have been updated to account for the changes related to its temperature increase. The air viscosity and density, however, have been computed based on the 373 K values for both idle and MTO. In the simulation, changes in viscosity and density to reflect

Table 5.3 – Fluid and thermal operating conditions

		Ground Idle	MTO	
Fluid				
	LP shaft speed	2,000	6,500	rpm
	HP shaft speed	10,000	15,000	rpm
	Bearing chamber abs. pressure	120	430	kPa
	Oil temperature for isothermal calculation	373		K
	Oil flow LP bearing (squeeze film damper)	0.037 (0.018)		kg/s
	Oil flow HP bearing (squeeze film damper)	0.050 (0.018)		kg/s
	Inlet air flow	9	50	g/s
	Scavenge pump vol. flow	18	28	l/min
	Scavenge ratio	2.42	3.77	[-]
Thermal				
	Outer wall temperature	373.6	463	K
	LP shaft temperature	381.5	535.8	K
	HP shaft temperature	390	475	K
	LP bearing oil inlet temperature (includes bearing heat generation)	380		K
	HP bearing oil inlet temperature (includes bearing heat generation)	390		K
	Sealing air inlet temperature	333	523	K

Table 5.4 – Material properties

		Ground Idle	MTO	
Fluid-Only Simulation				
Air density	ρ_a	1.1165	4.0102	kg/m ³
Air viscosity	μ_a	$2.21 \cdot 10^{-5}$		kg/(m·s)
Oil density	ρ_o	929.5		kg/m ³
Oil viscosity	μ_o	$4.65 \cdot 10^{-3}$		kg/(m·s)
Oil-air surface tension	σ_{o-a}	$2.45 \cdot 10^{-2}$		N/m
Thermal Simulation				
Air specific heat at constant pressure	$c_{p,a}$	1,013.27	1,037.24	J/(kg·K)
Air thermal conductivity	k_a	$31.46 \cdot 10^{-3}$	$41.11 \cdot 10^{-3}$	W/(m·K)
Oil specific heat at constant pressure	$c_{p,o}$	2,073		J/(kg·K)
Oil thermal conductivity	k_o	0.1278		W/(m·K)

the air inlet temperature of 523 K led to a progressive increase of the number of iterations required for convergence at each time step, and eventually to divergence after several weeks. The source of the divergence has been identified in the change of the air viscosity value. It is recommended for future simulations not to update the latter parameter, as its variation for bearing chamber conditions is limited ($2.21 \cdot 10^{-5}$ to $2.80 \cdot 10^{-5}$ kg/(m·s)), but rather update only the air density (4.0102 to 2.884 kg/m³). The latter option has proven feasible, although time constraints prevented to complete this set. The oil properties have been kept constant in all the simulations, because only small variations take place for the temperature range considered.

Comparing the two operating conditions of Table 5.3, the chamber has a large variability in terms of shaft speeds, pressurisation, and sealing air flows. The scavenge ratio is also variable between the two power settings, as it increases from 2.4 at idle to nearly 3.8 at MTO. Oil inlet flows are significantly higher than the KIT bearing chamber conditions simulated in Chapter 4: in this case, the total flow of oil is 123 grams per second, while in the KIT chamber it is approximately one fifth. For this reason, the oil quantity expected to accumulate in the BR725 chamber annulus is higher than the one in the KIT chamber, although the different scavenge design may overcome this effect.

The thermal operating conditions present also a large variability between the two engine power settings. In particular, the bearing chamber wall temperatures increase

significantly from idle to MTO, with variations of the order of 100 K or higher. This is due to the much higher heat generation of the engine associated with the take-off setting. In the oil system, this corresponds to a lower oil feed temperature upstream of the bearings (330 K at MTO against 360 K at idle [135]) to provide sufficient cooling. As the higher heat generation in the bearings at MTO offsets the lower feed temperature, it has been agreed with the industrial supervisors [136, 137] that the oil inlet temperatures in the CFD model should be selected equal for both idle and MTO. This choice brings the additional advantage of enabling to compare the oil heat pick-up in the two cases by simply looking at its temperature differences between the two simulations.

5.3.4 Boundary Conditions

With reference to Figure 5.4(a), the two shafts and the related bearing races are set as moving walls, with a non-slip shear condition; all the other walls are stationary. The air inlet flow has been assumed to equally split between the three seals, e.g. at idle the flow rate per each seal will be 3 grams per second. The tangential component of the air inlet flow has been computed as

$$U_{\perp a, in} = \frac{1}{2} u_{sh} \quad (5.1)$$

for the LP and HP shaft seals, and as

$$U_{\perp a, in} = \frac{1}{2} \left(\frac{u_{HPsh} + u_{LPsh}}{2} \right) \quad (5.2)$$

for the seal between the LP and HP shaft. In this case, u_{sh} is the shaft surface speed in proximity of the seal. The inlet turbulent intensity has been kept to the default 5%, and the length scale set equal to the seal channel thicknesses reported in Table 4.1.

In a different arrangement from the KIT bearing chamber, oil leaves the bearings on both sides in this model. Therefore, two inlet surfaces per bearing (front and rear) have been created, and an equal repartition of the flow between the front and rear side of each bearing has been assumed. The latter choice is consistent with the latest CFD observations of Adeniyi et al. [97], who found an equal axial repartition of the oil leaving the bearings in a bearing chamber. A degree of caution is required however, as the case analysed in [97] is a symmetrically under-cage lubricated bearing and may not fully reflect the BR725 RBC configuration. The oil flow entering from the bearing boundaries includes both the contribution of the jet and that of the squeeze film dampers. The inlet angle has been

selected to 45° for all four boundary surfaces, while the turbulence has been specified with intensity and viscosity ratio to values of 5% and 10 respectively.

The bearing chamber model includes a tank located under the sump, analogously to the KIT set-up. The bottom of the tank has a negative velocity boundary condition to match the required pump scavenge ratios. The vent pipe outlet has a pressure boundary condition instead, which is set to the chamber operating pressure depending on the engine power setting. Since little information is available on the turbulent characteristics of the air outlet flow, the backflow turbulent intensity of the vent has been kept to the default 5% with a viscosity ratio of 10. As for the KIT bearing chamber model, the effect of the surface tension between the oil and the air is accounted for by the Continuous Surface Force model of Brackbill et al. [126].

5.3.5 Solver Settings

VOF Model

Consistently with the settings adopted for the simulations in Chapter 4, the BR725 Rear Bearing Chamber case relies on the implicit formulation of the VOF model. This allows for good convergence also when regions with Courant number higher than unity are present in the domain (see Section 4.4.2). As in the KIT bearing chamber, these are located in proximity of the vent pipe inlet. The implicit body force formulation is active, as in the KIT model.

Turbulence Model

Based on the evaluation performed on the KIT bearing chamber in Chapter 4, the SST $k-\omega$ model with Egorov's turbulence damping [80] has been chosen for the BR725 bearing chamber simulation. The damping factor value selected is $B = 50$, as discussed at the end of Section 4.4.4.

Equations

The solver relies on a bounded second order implicit transient formulation, with coupled pressure-velocity equations and PRESTO! pressure discretization. Unlike the KIT model, the latter choice has proven more robust and faster converging than the body force weighted method. Momentum, turbulent kinetic energy, dissipation, and energy equations use second order upwind spatial discretization. The settings that differ from the described set-up are reported for each specific case in the Results sections.

Time stepping

The time step size is limited by the dimension of the mesh cells close to the sealing air inlets. This region corresponds to the maximum radius of both the HP and LP shafts, thus where their surface velocity is the highest. Cell sizes in this region are limited by the thin gap representing the seal; hence, the high speed/low cell size combination implies that a small time step size is required to model appropriately the flow in this region. The option of “stretching” the cells in the direction of the shaft rotation has been considered, but ANSYS Fluent does not allow this choice for a polyhedral mesh. An attempt to combine polyhedral and hexahedral cells has been also performed, but without a successful outcome due to the challenges encountered when trying to link the two mesh regions together.

Based on the seal gap limitations described above, the time step sizes for a maximum Courant number value of unity in proximity of the seals have been computed. As Table 5.5 reports, the cell length in the tangential direction $\Delta l_{\perp}$ limits the maximum time step size Δt_{max} , in particular for the HP shaft seal. As the computed time step size is too small to simulate several seconds of flow, it was decided to opt for larger time steps and accept Courant number values higher than one in some of the outer regions of the shafts.

Table 5.5 – Time step limits for $CFL_{max}=1$ near seals and final chosen time step

	Cell $\Delta l_{\perp}$ [mm]	Ground Idle Δt_{max} [s]	MTO Δt_{max} [s]
LP shaft seal	0.2	$1.3 \cdot 10^{-5}$	$4 \cdot 10^{-6}$
HP shaft seal	0.1	$1.3 \cdot 10^{-6}$	$8 \cdot 10^{-7}$
Time step chosen [s]		10^{-5}	$5 \cdot 10^{-6}$

The time steps selected are 10^{-5} seconds for idle and $5 \cdot 10^{-6}$ seconds at MTO. Figures 5.5(a) and (b) show the corresponding contour plots of the cell Courant numbers in proximity of the shaft surfaces. Regions with $CFL > 1$ cover the outer parts of the HP shaft (on the left hand side in the figures), with values of the order of 10 at idle and 5 at MTO. These regions affect only small portions of the shaft, while most of the surface has values below unity. For this reason, the chosen time step sizes have been considered appropriate for the simulation. The single-phase results of Section 5.4 later confirmed this assumption.

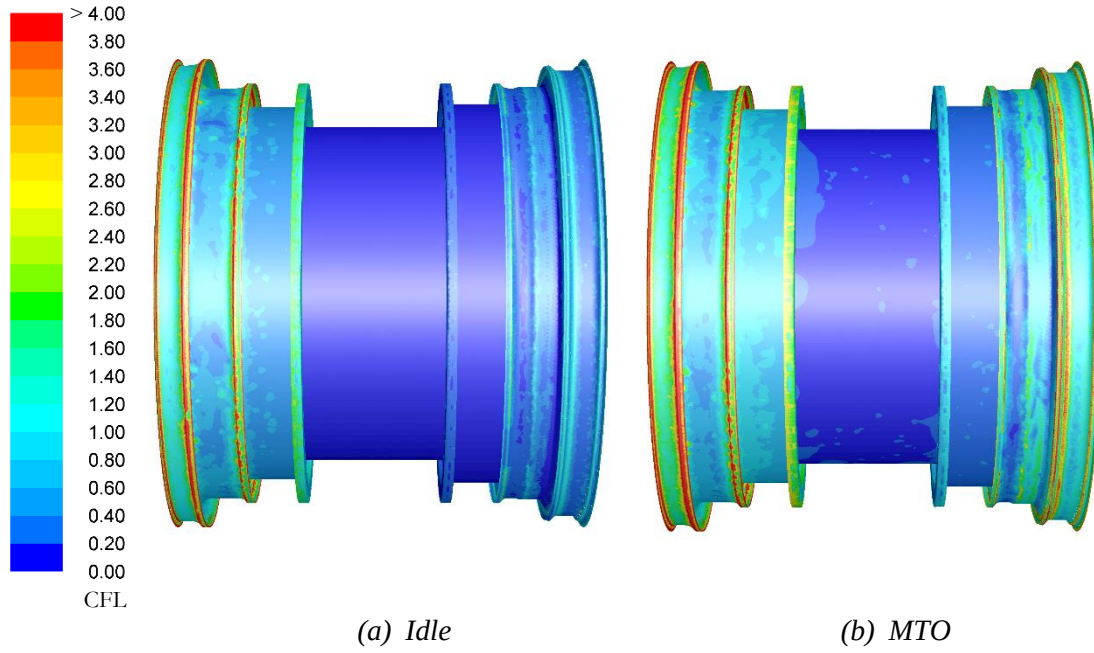


Figure 5.5 – Contours of cell convective Courant numbers (CFL) in proximity of the shaft surfaces; (a) $\Delta t=10^{-5}$ s, $CFL_{max}=26.37$, (b) $\Delta t=5 \cdot 10^{-6}$ s, $CFL_{max}=10.25$

Interface Reconstruction

Based on the simulations performed on the KIT bearing chamber, the Compressive interface reconstruction scheme is adopted for all cases in this chapter. In addition to fast convergence times, the scheme also ensures robustness of the simulations, as these cope well with discontinuous changes in the operating conditions of the chamber. In other words, it is possible to perform the switches between different shaft speeds and fluid densities using only one or no intermediate step while maintaining convergence (see Section 5.3.6).

Initialisation

The flow initialisation has been performed by letting the oil ‘drip’ from the bearing inlets without shaft rotation and air injection, following the procedure reported by Peduto et al. [41]. This initial phase took about 28 hours of computation to complete, but enabled the simulation to start from a steady, realistic flow condition. As the tank at the bottom of the chamber should remain filled with oil while performing the simulation, regular patching of the liquid phase in this region is required.

5.3.6 Simulation Strategy

In order to assess the impact of the different parameters affecting the flows in the bearing chamber, and ensure the simulation stability, a methodology based on individual

changes of the boundary conditions has been defined. The strategy aims at increasing the complexity of the flows to be resolved as the simulation progresses. The first step is to obtain the initialisation as described in the previous section. In this case, no shaft rotation or sealing air are introduced, so the stationary state can be achieved rapidly with a relatively large time step (10^{-3} seconds). The second step is the introduction of the shaft rotation without sealing air injection, allowing the oil to spread on the bearing chamber walls. This is done by using half of the idle rotational speeds for a short time interval, prior to switching to idle. At this point, once the oil residence volume is stable, sealing air is injected, thus setting the third phase of the simulation. Upon the achievement of the stationary state, the heat transfer modelling is activated until the average annulus temperature is stabilised. The MTO simulation starts from this latter condition. Heat transfer modelling is deactivated and the operating conditions changed. Once the flow stationary state is achieved, the energy equation is again activated until this final simulation is complete. The simulation strategy is summarised in the schematic of Figure 5.6.

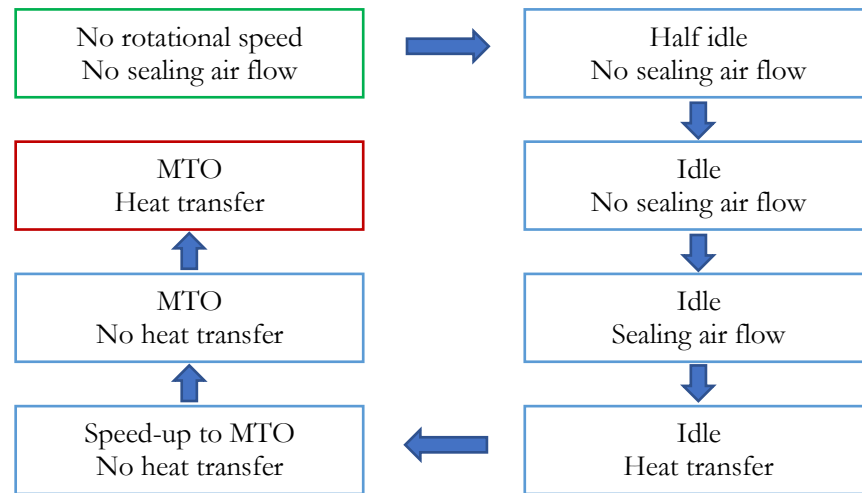


Figure 5.6 – BR725 RBC simulation procedure

5.3.7 Set-up Summary

Table 4.3 summarises the settings adopted for the BR725 bearing chamber simulation. Additional information on the operating conditions and fluid properties is available in Tables 5.3 and 5.4. The settings of the single-phase mesh independence study are reported in Section 5.4.

Table 5.6 – BR725 RBC two-phase simulation settings

Multiphase Model	Volume of Fluid (VOF)		
Turbulence Model	SST k- ω		
VOF Turbulence Damping	Egorov [80]		
Damping Factor	50		
Phase Interaction	Surface tension with CSF model [126], $\sigma_{o,a} = 0.0245$ N/m		
Boundary Conditions		Idle	MTO
	Oil	$\dot{m}_{o,in} = 4 \times 30.75$ g/s	Inlet angle $\gamma = 45^\circ$
	LP shaft seal	$\dot{m}_{a,in} = 3$ g/s $U_{\perp a,in} = 7.87$ m/s	$\dot{m}_{a,in} = 16.67$ g/s $U_{\perp a,in} = 25.59$ m/s
	LP-HP shaft seal	$\dot{m}_{a,in} = 3$ g/s $U_{\perp a,in} = 19.23$ m/s	$\dot{m}_{a,in} = 16.67$ g/s $U_{\perp a,in} = 34.31$ m/s
	HP shaft seal	$\dot{m}_{a,in} = 3$ g/s $U_{\perp a,in} = 39.38$ m/s	$\dot{m}_{a,in} = 16.67$ g/s $U_{\perp a,in} = 59.07$ m/s
	Vent	$p_{gauge} = 0.2$ bar	$p_{gauge} = 3.3$ bar
	Sump	$U_{o,out} = 0.038225$ m/s	$U_{o,out} = 0.059463$ m/s
	LP shaft rot. speed	2000 rpm	6500 rpm
	HP shaft rot. speed	10000 rpm	15000 rpm
Time step Size		10^{-5} s	$5 \cdot 10^{-6}$ s
Pressure-Velocity Coupling	Coupled		
Gradient	Least squares cell based		
Pressure Discretization	PRESTO!		
Momentum Discretization	Second order upwind		
Turbulent Kinetic Energy Discr.			
Specific Dissipation Discr.			
Body Force Formulation	Implicit body force		
Volume Fraction Formulation	Implicit		
Interface Computation	Compressive		
Transient Formulation	Bounded second order implicit		
Mesh Size (Cells)	$9.87 \cdot 10^6$		
Initialisation	Oil dripping from bearings with no shaft rotation/sealing air		

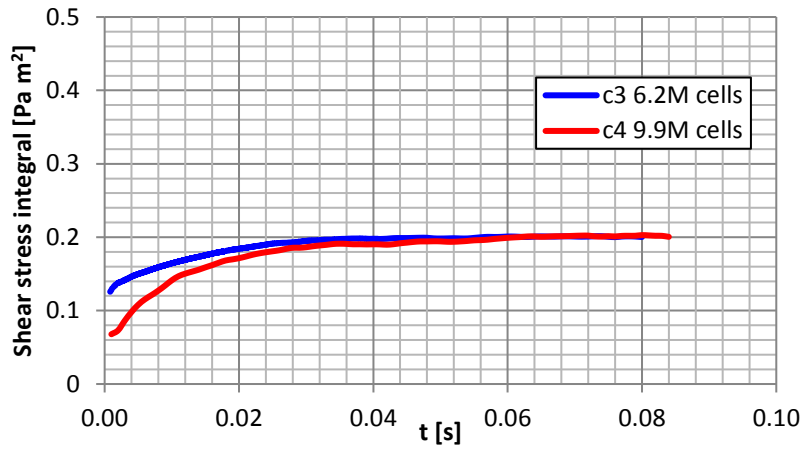
5.4 Single-Phase Results

Single-phase, air-only simulations have been carried out as a preliminary assessment to evaluate the robustness of the model. The analysis compares the shear stress integrals on the chamber outer wall and on the two shafts at idle, for three different meshes: 6.2, 7.9, 9.9 million cells. The simulations have been run until the shear stress integrals achieved time independence. The settings are the same as those presented in Table 4.3 for idle conditions, excluding the multiphase modelling options.

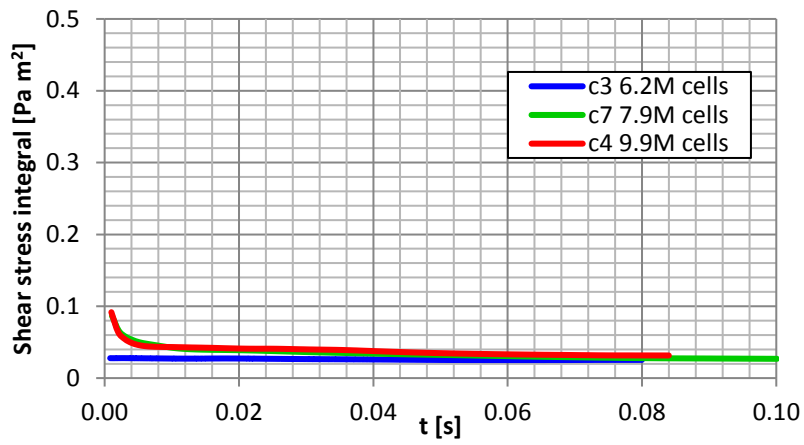
As shown in the plots of Figure 5.7, mesh independence for the air shear stress integrals is achieved with the 7.9 million cells mesh. However, the final choice for the VOF simulation is the 9.9 million cells mesh, because of its more regular cell size distribution in the regions far from the walls.

The results suggest that the stability of the shear stress integral is more difficult to achieve on the HP shaft surface as opposed to the remaining walls, as in the upper range of its diameter the Courant number exceeds one for the chosen time step (see Figure 5.5 in Section 5.3.5). Despite the Courant number higher than unity in part of the HP shaft surface, the implicit VOF formulation ensured quick convergence at each time step.

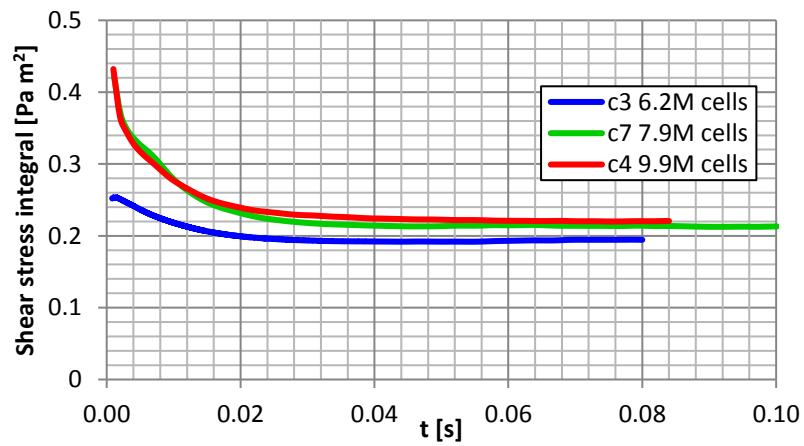
The single-phase results confirmed the capability of the numerical model to run flawlessly through the transient simulation. This represents a solid starting point for the implementation of the VOF method.



(a) Outer wall shear stress integral



(b) LP shaft shear stress integral



(c) HP shaft shear stress integral

Figure 5.7 – Time-evolution of shear stress integrals from mesh independence study at ground idle conditions

5.5 Two-Phase Results

5.5.1 Oil Residence Volume

The oil residence volume is measured in the annular region of the bearing chamber, i.e. in the region of radius $r < 99.0 \text{ mm}$ and length $l = 129.0 \text{ mm}$, which excludes the scavenge ramp. The plot of Figure 5.8 reports the time evolution of the oil residence volume through all the operating conditions simulated, following the steps listed in Section 5.3.6. The transitions between operating conditions have been decided upon the stabilisation of the oil residence volume at each stage.

The time evolution plot of the residence volume shows that the amount of oil in the bearing chamber annulus starts increasing from its 108 millilitres equilibrium point at 0 rpm as the shafts start rotating at the idle speed. This is due to the oil spreading on the whole bearing chamber outer wall surface because of the air shear generated by the rotation of the shafts. This effect gives a 35% increase in oil amount in the chamber annulus.

Once sealing air is injected, the oil residence volume decreases as a pressure gradient of about 10 kPa establishes between the chamber annulus and the vent pipe. This generates a suction effect on the oil in proximity of the vent pipe inlet, which is clearly visible in the animations of the time evolution of the oil surface near the vent. The effect of sealing air

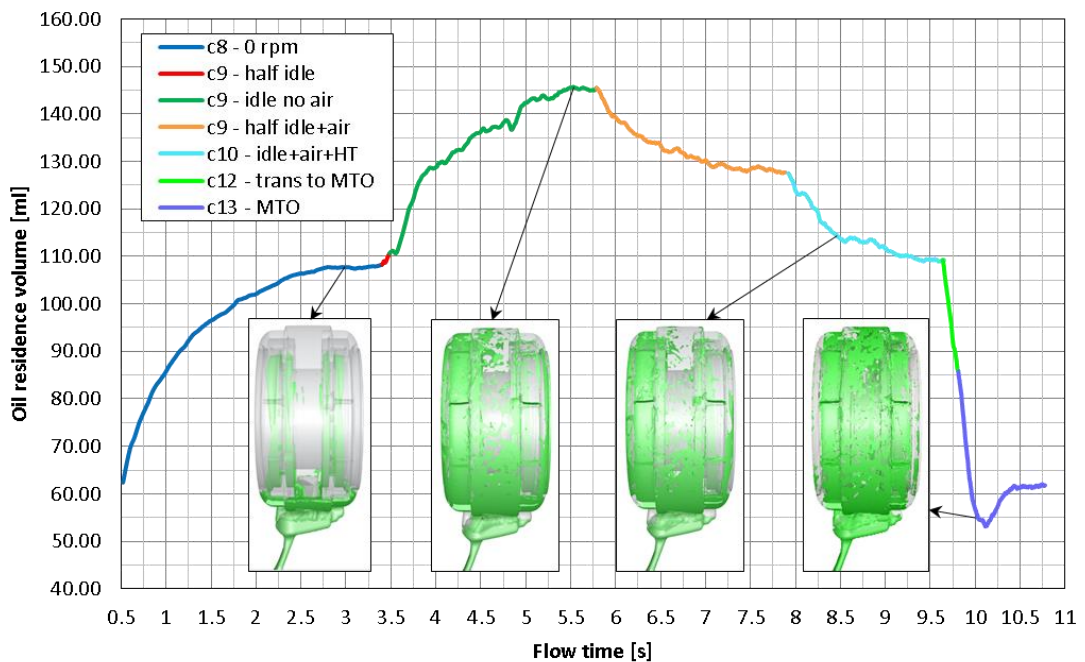


Figure 5.8 – Time-evolution of oil residence volume in bearing chamber annulus

reduces the oil residence volume to about 110 ml at idle, close to the amount observed at 0 rpm.

As the two shaft rotational speeds, sealing air rate, and chamber pressure are increased to meet the MTO conditions, the oil residence volume decreases sharply to about 62 ml. The major contributor to this effect is the 5.5 times increase in sealing air flow, which leads to a pressure difference of about 35 kPa between the main chamber annulus and the vent pipe, enhancing the oil suction through the latter. To better visualise the different pressure conditions taking place at idle and maximum take-off, Figure 5.9 shows for both conditions the contour plots in a cross-section of the annulus including the vent pipe.

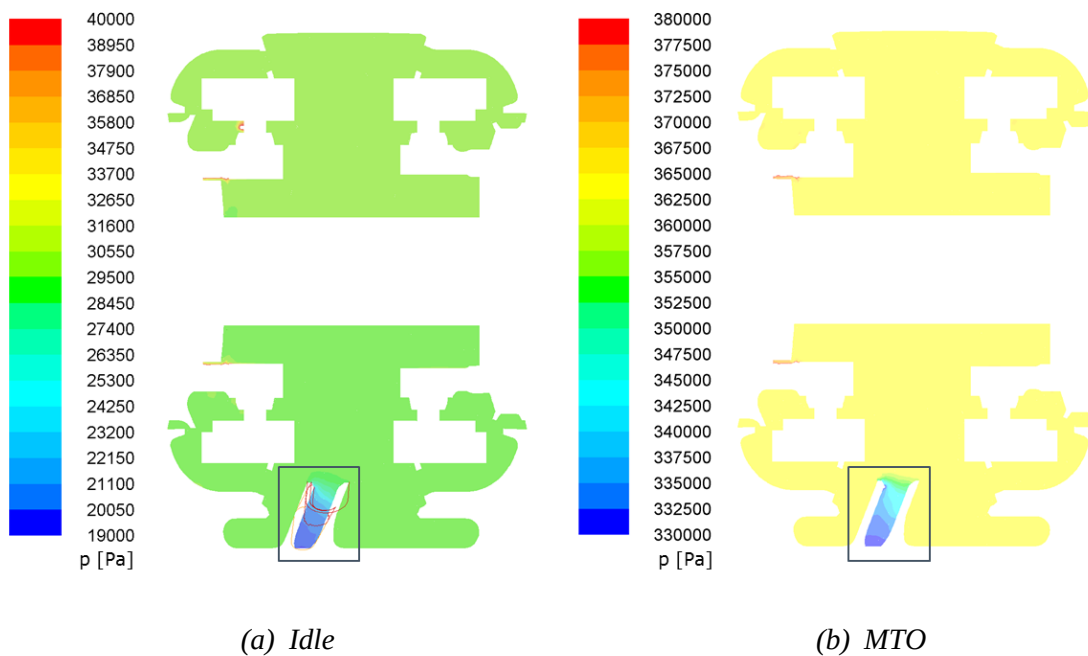


Figure 5.9 – Contours of static pressure in bearing chamber cross-section including vent pipe.
(a) Flow time = 8.48 s; (b) Flow time = 10.23 s

5.5.2 Oil Outlet Flows and Scavenge Efficiency

The oil flows leaving the chamber through the scavenge and vent pipes have been monitored throughout the whole simulation. In the scavenge pipe, the monitor plane is located downstream of the sump, while the vent pipe has the plane located at its outlet, as shown in Figure 5.10.

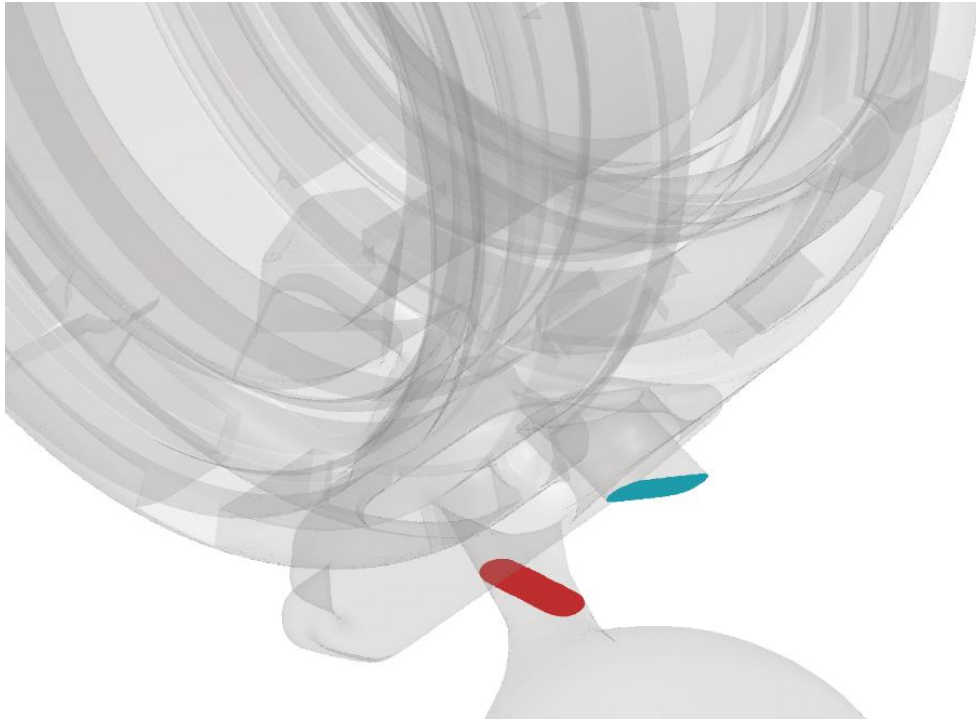


Figure 5.10 – Location of monitor planes for oil outlet flows: scavenge pipe (red) and vent pipe (blue)

Figures 5.11 and 5.12 report the time evolution of the oil flow through the scavenge and vent pipes for the different operating conditions simulated. At idle, the plots of the outlet flow rates show that the beginning of the injection of sealing air has a significant impact on the amount of oil leaving the chamber through the vent pipe. While the scavenge flow keeps constant in the range between 100 and 140 g/s, the oil flow rate in the vent increases sharply from nearly zero to about 30 g/s. The near-zero value indicates that oil tends to flow around the protruding vent as long as there is no air going through the latter. Once a flow of air is established, oil surrounding its inlet tends to entrain with it, leading to the decrease in oil residence volume observed in Section 5.5.1.

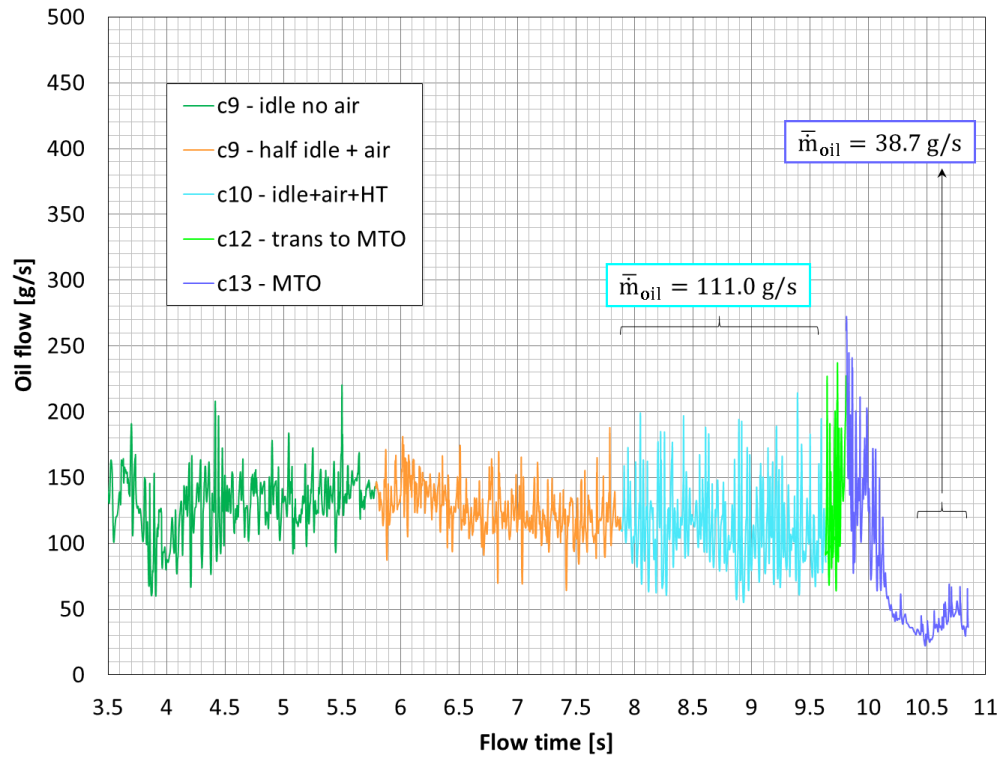


Figure 5.11 – Time-evolution of oil flow rates in scavenge pipe and computed average values

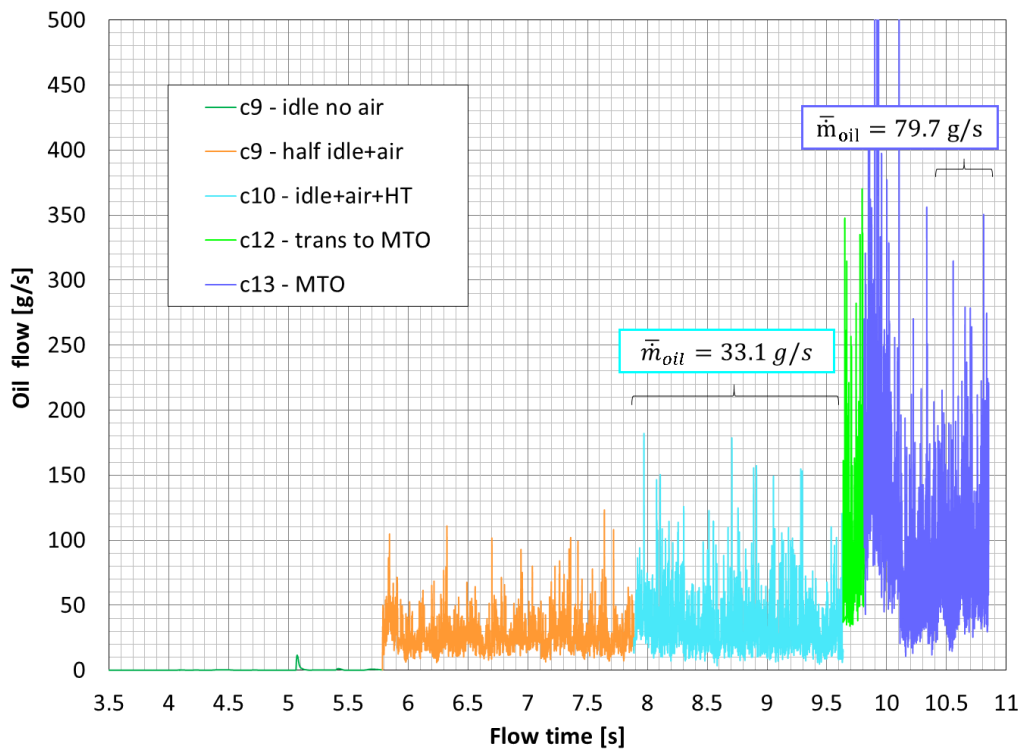


Figure 5.12 – Time-evolution of oil flow rates in vent pipe and computed average values

The effect of sealing air on the outlet oil flows reaches its peak at MTO conditions. In this case, the air flow increases from 9 g/s to 50 g/s, generating a much stronger suction

effect in the region surrounding the vent pipe inlet. The oil flow rate in the vent reaches a final average of about 80 g/s, i.e. about 65% of the oil feed. This affects significantly the scavenge efficiency, which drops from 90.2% at idle to 31.5% at MTO. While the idle result is aligned with the design assumptions, which forecasted a value of about 80% in these conditions [136], the MTO value is lower than expected, as the design assumptions are in the range of 50% [136]. A contributor to this discrepancy is likely to be the gas-liquid interaction modelling, which for MTO conditions tends to over-predict the effect of air shear at the vent, as discussed in Chapter 4.

Although test data from the BR725 does not cover the rear bearing chamber outlet oil flows, results from the tests performed on the BR700NG provide a good reference for comparison, as the two rear bearing chamber designs are similar [136]. In the BR700NG test results, the scavenge efficiency is 93% at idle and varies between 66% at 90% MTO speed and 47% at 95% MTO speed [136]. At idle, the sealing air flow of the BR700NG is 8 g/s, thus very close to the 9 g/s of the BR725. Therefore, in this condition, the CFD result of 90.2% is validated with good accuracy, indicating that VOF is a suitable method to predict the scavenge efficiency at idle.

The comparison of the MTO case has a higher degree of uncertainty, as the sealing air flow is 35 g/s at 95% MTO in the BR700NG against the 50 g/s at 100% MTO in the BR725. The scavenge efficiency of 47% measured at 95% MTO suggests that this value will be lower when at 100% MTO, thus closer to the 31.5% found with the CFD analysis. This indicates that the discrepancy between the CFD results and the real flow conditions may be lower than the 18.5% based on the design assumption. In this perspective, although VOF certainly performs less accurately at MTO than at idle, the levels of predicted scavenge efficiency do not deviate excessively from the test flow conditions. The latter finding also confirms that the selection of the turbulence damping factor value for this case, based on the KIT bearing chamber experience, was a sensible choice.

5.5.3 Frequency Content of Oil Outlet Flows

Additionally to providing information on the oil split between the vent and scavenge pipes, the data on the time evolution of the outlet flows has been analysed to extract their frequency content. For this purpose, a Fast Fourier Transform (FFT) of the data reported in the plots of Figure 5.11 and 5.12 has been computed. The analysis considers 1024 samples of the oil outlet flows at ground idle and MTO. The sampling frequency is 1000

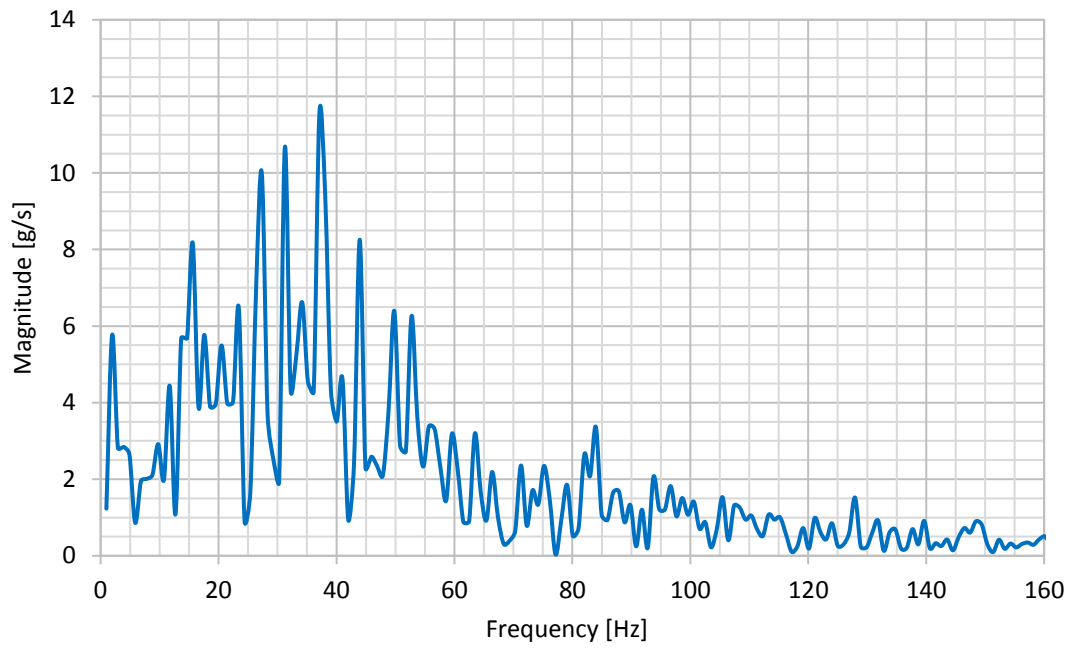
Hz for the idle data and 2000 Hz for MTO, with time intervals covering 1.023 and 0.511 seconds respectively. Table 5.7 summarises the sampling set-up.

Table 5.7 – FFT sampling characteristics

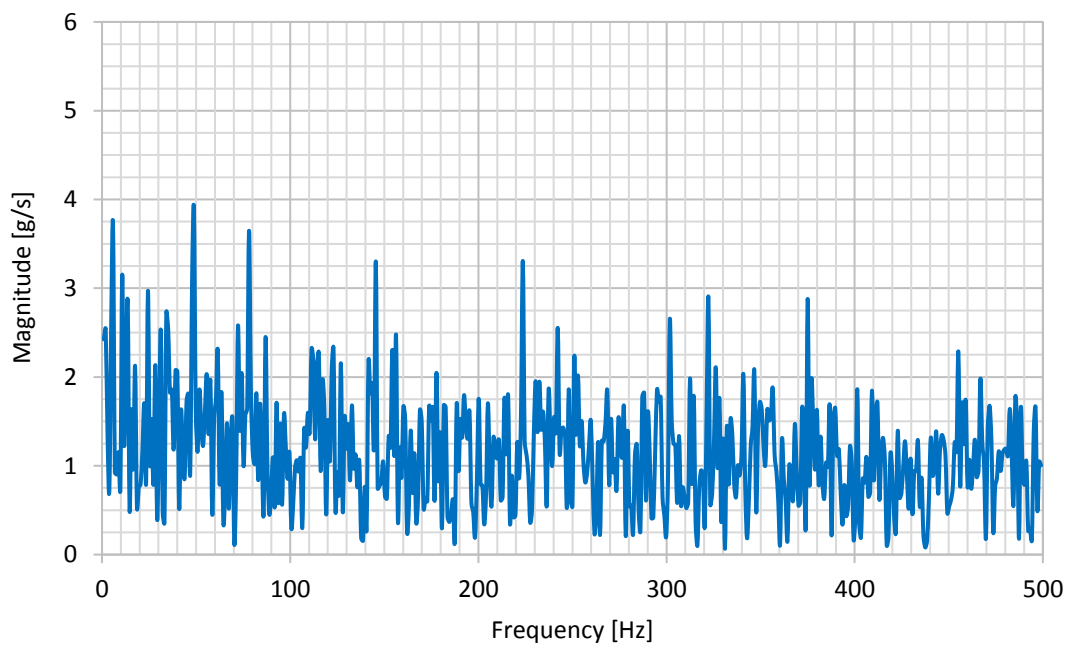
		Ground Idle	MTO
Samples analysed	[-]	1024	1024
Sampling frequency	[Hz]	1000	2000
Sampling interval	[s]	8.631-9.654	10.340-10.850

The results of the analyses are plotted in Figures 5.13 and 5.14. A clear pattern of dominant frequencies is found in the scavenge flow, both at ground idle and maximum take-off power. In the first operating condition, the largest oscillations in oil flow rate are in the range between 15 and 55 Hz, with a maximum peak of 12 grams per second at 37 Hz. The MTO power setting has smaller oscillations instead, i.e. of the order of 2 grams per second between 10 and 45 Hz. These patterns cover a range of frequencies comparable with those found by Peduto et al. [41], who identified an interval of oscillations between 2 and 40 Hz for the oil film thickness signal in proximity of the scavenge.

Although the results of Peduto et al. [41] are for a shaft rotational speed of 3500 rpm, a link is identifiable between the patterns of the oil film thickness oscillations and the scavenge oil flow rate. In fact, a decrease in oil film thickness next to the scavenge inlet implies that, if oil flows towards the latter, an increase in flow rate through the scavenge pipe will be detected. Extending this reasoning to periodic oscillations, the variations in film thickness and consequent scavenge flow rates will tend to have the same frequency, as the waves propagate through the scavenge pipe. Analogously, the amplitude of the waves in the scavenge flow will tend to reflect that of the film thickness variations. Therefore, the analysis of the frequency content and amplitude in the oil outlet flow can provide an insight in the flow conditions upstream of the scavenge pipe inlet. Large oscillations imply that oil tends to accumulate in large ‘lumps’ prior to leaving the bottom of the chamber, which is a condition typical of gravity-dominated flows. Smaller oscillations, such as those found at MTO in Figure 5.14(a), will tend to represent shear-dominated flows, where large accumulations of oil upstream of the scavenge cannot take place due to the tendency of air to generate a smooth, thin film. Therefore, the analysis of the outlet flow patterns at the scavenge can provide a first indication on whether the flow inside the chamber will behave as gravity- or shear-dominated.

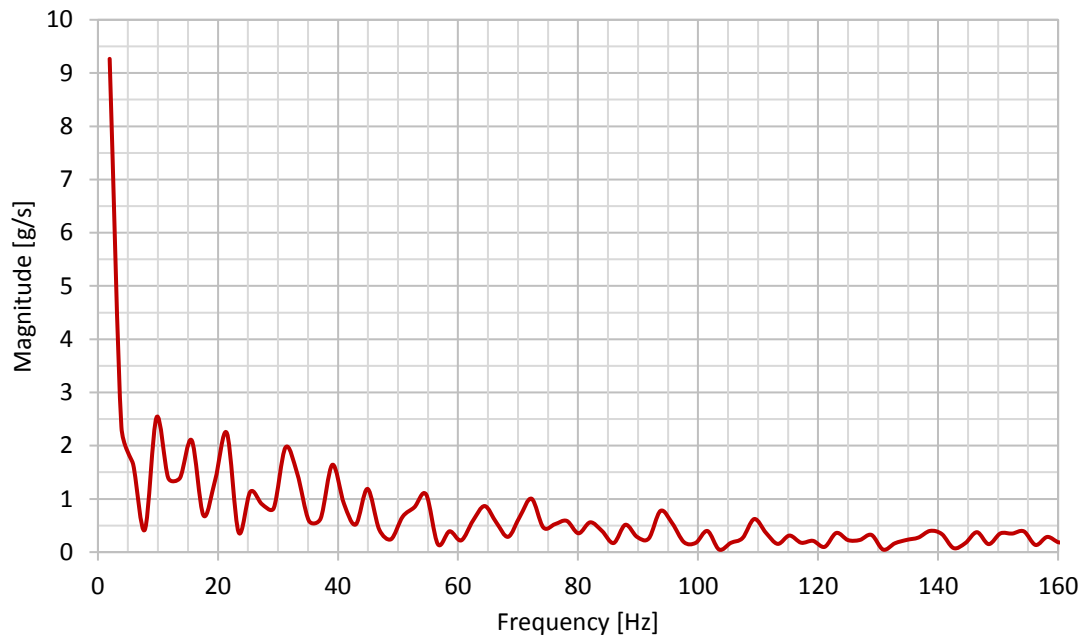


(a) Scavenge oil flow

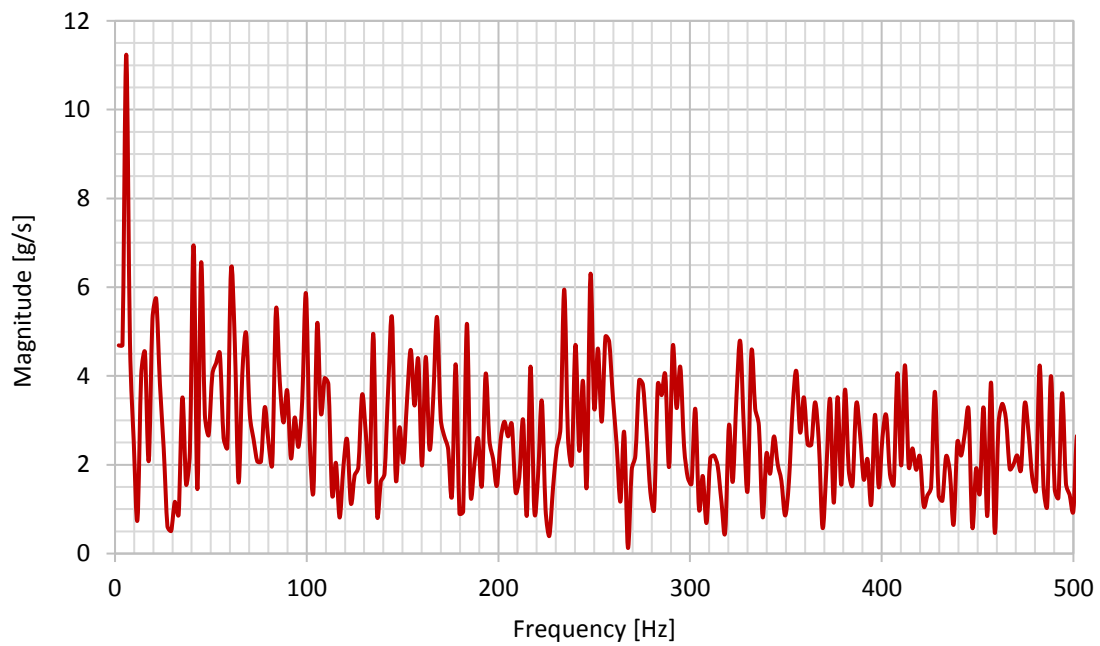


(b) Vent oil flow

Figure 5.13 – Frequency content (FFT) of oil outlet flows at ground idle



(a) Scavenge oil flow



(b) Vent oil flow

Figure 5.14 – Frequency content (FFT) of oil outlet flows at MTO

Considering the oil flow rates in the vent pipe, no dominant frequencies are found at either ground idle or MTO conditions. This behaviour is linked with the ingestion of low oil volume fractions diffused in the air flow. Combined with the high air velocities in the vent, i.e. of the order of 100 m/s, the oil outlet behaviour in this pipe tends to be randomised and not to reflect any periodic condition inside the chamber. Nonetheless, some information can be extracted from the amplitude of the oscillations, as they are larger at maximum take-off than at ground idle, consistently with the larger oil flow leaving the chamber through the vent in the MTO condition.

5.5.4 Oil Film Velocities

Oil film velocities have been evaluated by plotting the contours of the flow tangential velocities on the iso-surfaces of oil volume fraction $\alpha_o = 0.5$. Figure 5.15 reports the results for ground idle and MTO conditions, plotted on the co-current and counter-current side of the chamber.

The contours at idle show that the film surface velocity is generally less than 2 m/s in the co-current side, with the higher values in the bottom part of the chamber. This is due to the contribution of gravitational acceleration, which acts on the film as it flows down towards the scavenge. In the counter-current side, oil tends to flow back towards the bottom of the chamber at less than 0.4 m/s. Upward-directed films on this side of the chamber are found only in proximity of the HP shaft, i.e. on the left-hand side of Figure 5.15(b). In this region, the air shear is sufficiently strong to drive the oil towards the top of the chamber.

Considering the MTO operating condition, the effect of gravity on the film velocity is clearly identified in the co-current side of the chamber, where peaks of up to 3 m/s are found in small regions on the lower part of the outer wall and in proximity of the scavenge ramp. On the counter-current side, in the central region of the chamber annulus oil tends to flow back towards the bottom of the chamber, with the same low velocity found in the idle setting. The difference with the latter condition is a higher coverage of the wall surface in the top part of the chamber, as well as a co-current flow condition in both axial extremities, i.e. in the slots between the bearing supports.

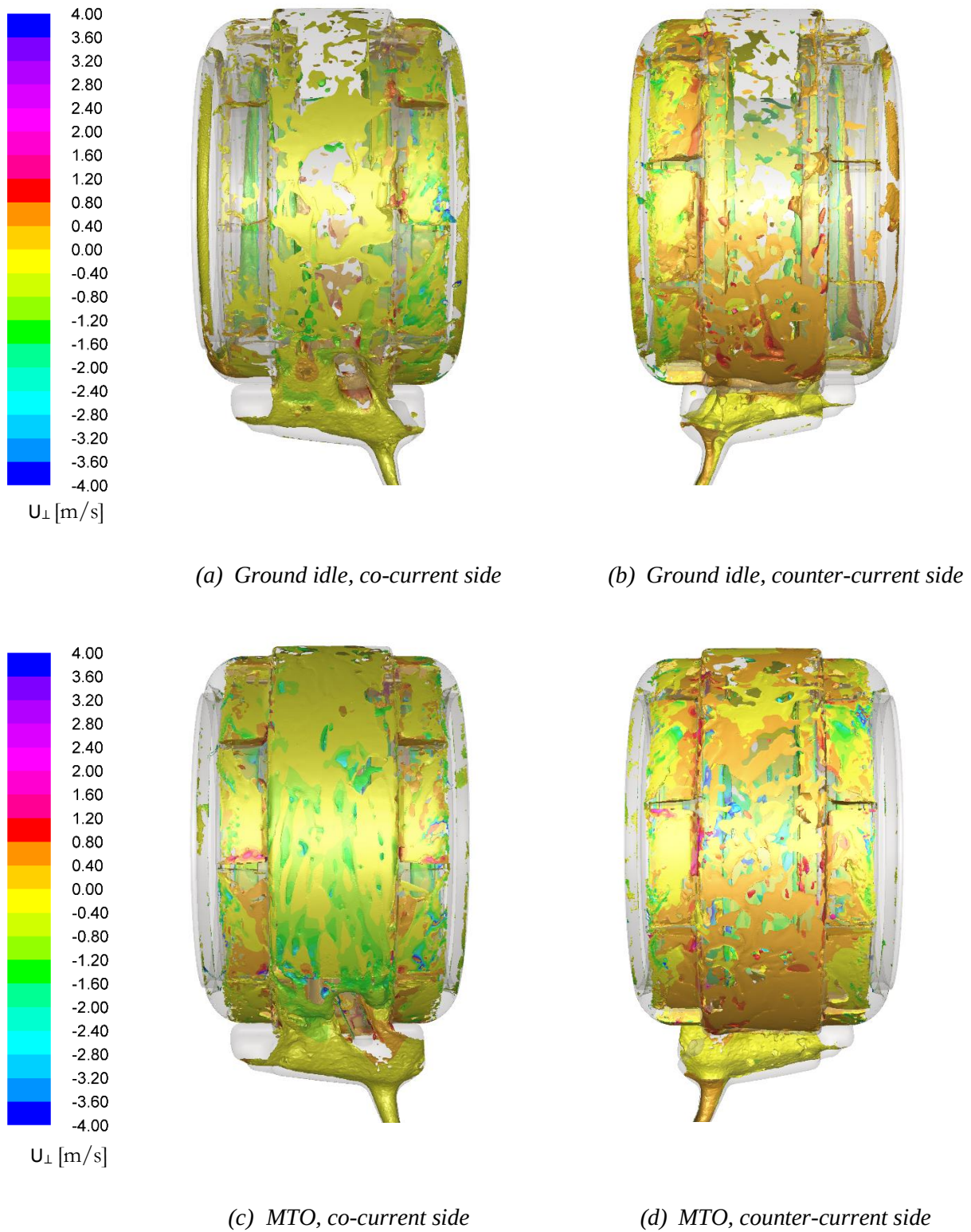


Figure 5.15 – Contours of oil film surface tangential velocities at $\alpha_o = 0.5$.
Co-current flow for $U_I < 0$. (a,b): Ground idle, $t=9.635$ s; (c,d): MTO, $t=10.847$ s

The co-current flow behaviour at both axial extremities of the chamber confirms the increased contribution to the film momentum given by the faster shaft rotation. However, this appears insufficient to keep the film flowing upwards in the central region of the annulus. A contributor is the presence of the jet block, which generates a recirculating air flow driving the oil towards the scavenge, as shown in Figure 5.16. The result is a film flowing counter-current and at low velocity. From a thermal management point of view, this may lead to oil overheating due to the prolonged contact with the wall surface.

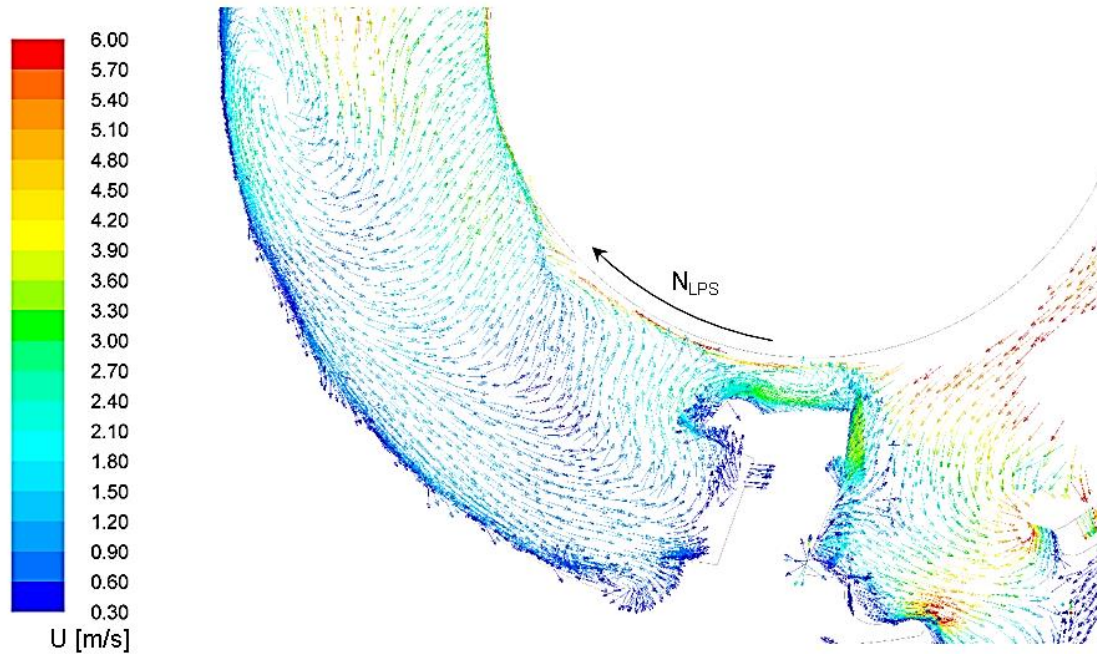


Figure 5.16 – Visualisation of recirculation zone generated by jetblock at MTO, $t=10.85$ s

5.5.5 Film Thickness Distribution

Consistently with the approach followed for the KIT bearing chamber (Section 4.2.7), film thickness measurement areas have been defined at discrete locations around the annulus of the BR725 Rear Bearing Chamber. As the geometry is more complex than the previous case, additional care was needed for an appropriate positioning of the sensors. This included avoiding the inner struts of the bearing supports, as well as other features such as the scavenge ramp and the jet block. To help understand the positioning of the film thickness numerical ‘sensors’, a schematic of the bearing chamber annulus is provided in Figure 5.17.

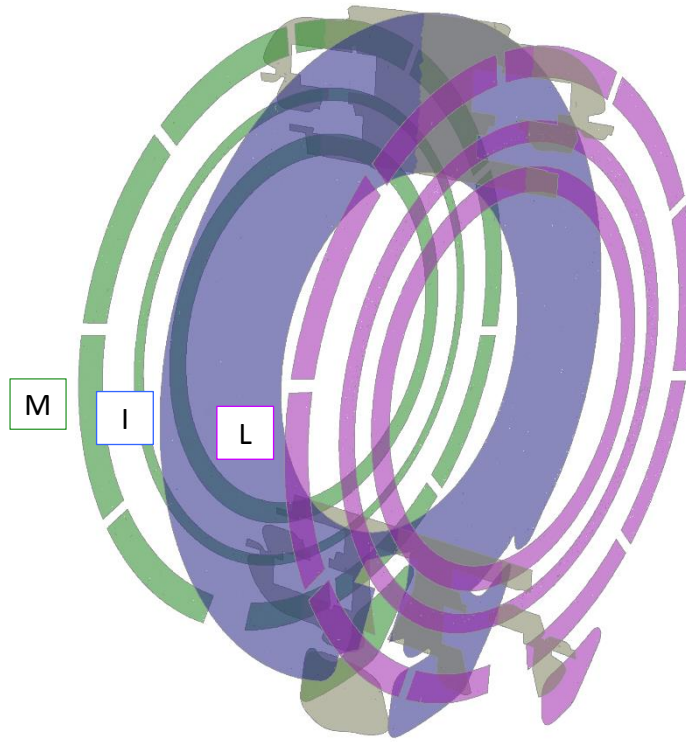


Figure 5.17 – Location of planes for oil film thickness computation

The film thickness is measured in the three planes shown in Figure 5.17, named ‘M’, ‘I’, and ‘L’, which correspond respectively to the HP shaft bearing location, centreline of the chamber, and LP shaft bearing location. In particular, sections ‘M’ and ‘L’ have been created to measure the oil film thickness within the slots between the bearing supports. While 30 measurement areas are assigned for each of the latter two sections (3 ‘sensors’ per slot), 26 measurement areas are used for the mid-section of the chamber. The lower number here is due to the geometrical features which prevent the measurement on a circular geometry, i.e. the scavenge ramp, the protruding vent, and the jet block. For this reason, the interval $130^\circ < \varphi < 200^\circ$ in section ‘I’ is not covered by the measurement areas. Consistently with the KIT bearing chamber post-processing, the oil film thickness has been computed from time-averaged iso-surfaces of $\alpha_o = 0.5$, with square measurement areas having side length of 5 millimetres. Figures 5.18 and 5.19 report the oil film thickness profiles for the two operating conditions simulated.

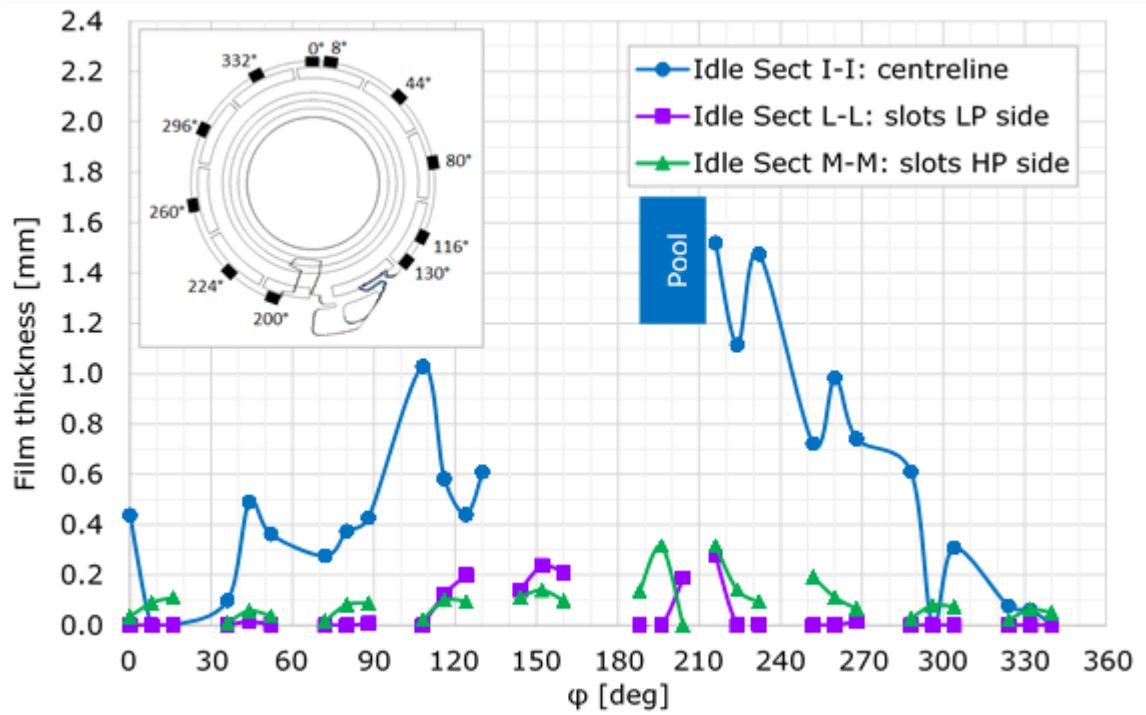


Figure 5.18 – Idle oil circumferential film thickness distributions, time-averaged over 7 samples, $t=9.185-9.635$ s

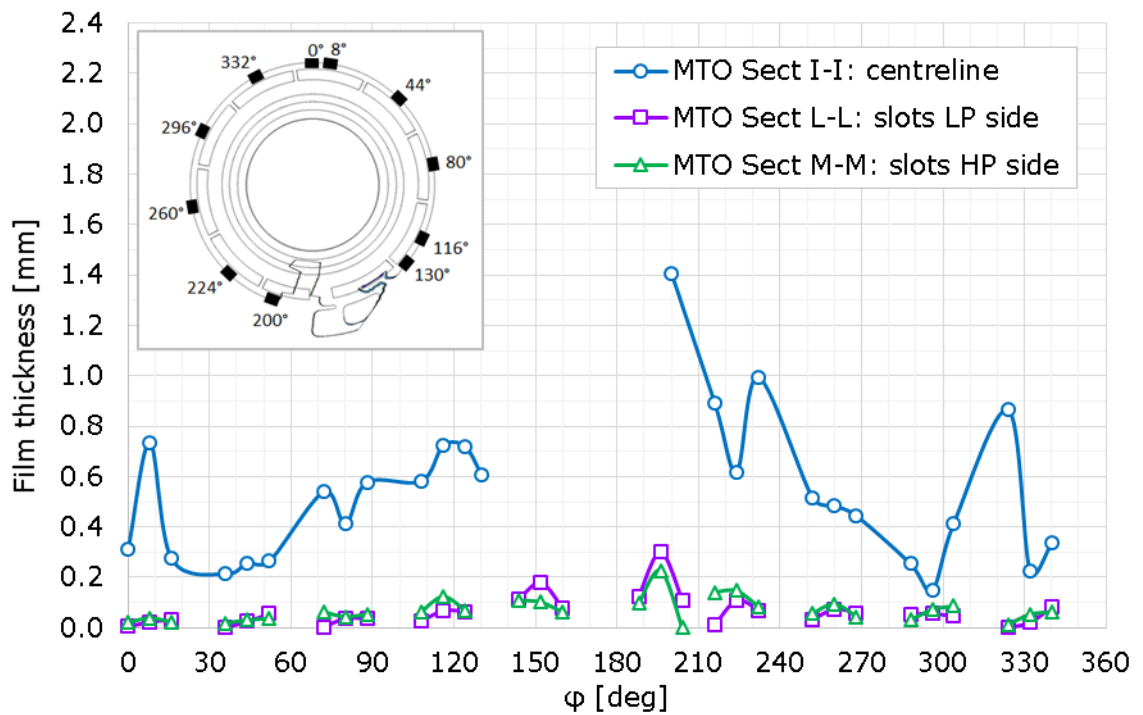


Figure 5.19 – MTO oil circumferential film thickness distributions, time-averaged over 6 samples, $t=10.560-10.847$ s

Considering the centreline section of the bearing chamber ('I') of Figure 5.18, the oil film thickness increases when moving along the co-current side towards the bottom of the chamber, i.e. from $\varphi = 0^\circ$ to $\varphi = 130^\circ$. The values range from less than 0.4 mm at the top to 0.6 mm towards the bottom of the chamber, with a peak of 1 mm at $\varphi = 108^\circ$. The accumulation of oil at the bottom of the co-current side is linked with the relatively low flow velocity, as seen in Section 5.5.4, which leads to an increase in film thickness when approaching the scavenge ramp. This behaviour is typical of gravity-driven flows, and finds matching results in the reference experimental measurements [10, 22] at the lowest shaft speed (5,000 rpm). On the counter-current side, a large pool is found at $\varphi = 200^\circ$. The film thickness on this sensor reaches a value of about 13 mm, hence is not reported in the plot. The presence of a pooling region next to the jet block has been confirmed by the observation of the oil behaviour in this area through animations, and is consistent with the gravity-driven nature of the flow, which leads to large accumulations of oil at the bottom of the chamber and near-dry walls at the top. The latter aspect is confirmed by the low film thicknesses observed in the 330° - 40° region in the plot of Figure 5.18.

The film thickness distributions computed in the slots ('L' and 'M') at idle show that oil tends to accumulate in their bottom part. In fact, the majority of the curves in the co-current side has a rising trend, while the opposite is found counter-current. This behaviour indicates that gravity has a relevant effect in defining the film thickness distribution within the slots. The struts act as accumulation surfaces for the oil, although the accumulations are rather limited, with maximum average film thicknesses of about 0.3 millimetres.

The MTO operating condition yields a smoother oil film than the idle case. This is identifiable in the centreline plot of Figure 5.19, where lower film thickness peaks are found. In particular, the counter-current side is not characterised by a pooling region at $\varphi = 200^\circ$, but by an oil film about 1.4 mm thick. A more uniformly distributed coverage of oil is found on this side of the chamber, consistently with the higher air shear forces which tend to push it towards the top of the annulus.

The MTO film thickness profiles in the centreline have trends similar to those found by Kurz et al. [10] for the 10,000 rpm covered ramp case. The film thickening in the rising side of the chamber is not found in [10] for the highest shaft speed (15,000 rpm), consistently with a highly sheared flow. The better match of the results with the 10,000 rpm operating condition in [10] rather than the MTO-representative 15,000 rpm suggests

that the different layout between the single-shaft experimental set-up [10] and the two-shaft BR725 Rear Bearing Chamber plays a role on the final flow regime. In the latter, the largest moving surface in contact with the fluid is the LP shaft, which has a surface area of 0.070 m^2 against the 0.032 m^2 of the HP shaft. Therefore, despite the higher rotational speed, the HP shaft is not deemed to have an influence as strong as the LP shaft on the oil flow behaviour. Nonetheless, the uncertainties related with the choice of the turbulence damping factor, which may be too low to identify a fully shear-driven flow, are still present.

5.6 Heat Transfer

5.6.1 Set-up

The boundary conditions for the setting of the thermal simulation have been obtained from the Rolls-Royce thermo-fluids model and the engine validation tests. While the in-house model gave the chamber wall and oil inlet temperatures, the test results provided the air inlet temperatures. Temperature-dependent fluid properties and wall thicknesses were originally accounted, but initial tests highlighted stability issues that prompted to simplify the problem by using constant fluid properties and not simulating heat conduction in the walls. For analogous stability reasons, the wall temperatures from the thermo-fluids model have been averaged, so that a single value is assigned to each boundary of the CFD model, without adding discontinuities by dividing it into multiple zones.

On top of enhanced stability of the simulation, using the simplest possible thermal set-up gave the advantage of assessing the minimum cost required to include heat transfer in the CFD model. The chosen boundary conditions and fluid properties, presented in Tables 5.3 and 5.4, are recalled in Table 5.8. The thermal model has been set-up analogously to the KIT bearing chamber, thus with the activation of the energy equation and the implementation of the additional boundary conditions and material properties listed in Table 5.8. Second order upwind discretization has been selected for the energy equation. The initialisation relies on patching the fluid temperature of the domain to the air inlet temperature. The latter phase fills most of the bearing chamber volume and is rapidly expelled, having little time to exchange heat with the oil or the chamber walls. The monitor selected to assess the achievement of the thermal stationary state is the average fluid temperature in the bearing chamber annulus.

Table 5.8 – Thermal boundary conditions and material properties

	Ground Idle	MTO	
Boundary Conditions			
Outer wall temperature	373.6	463	K
LP shaft temperature	381.5	535.8	K
HP shaft temperature	390	475	K
LP bearing oil inlet temperature (includes bearing heat generation)	380		K
HP bearing oil inlet temperature (includes bearing heat generation)	390		K
Sealing air inlet temperature	333	523	K
Fluid Properties			
Air density	1.1165	4.0102	kg/m ³
Air viscosity	2.21·10 ⁻⁵		kg/(m·s)
Air specific heat at constant pressure	1013.27	1037.24	J/(kg·K)
Air thermal conductivity	31.46·10 ⁻³	41.11·10 ⁻³	W/(m·K)
Oil density	929.5		kg/m ³
Oil viscosity	4.65·10 ⁻³		kg/(m·s)
Oil specific heat at constant pressure	2073		J/(kg·K)
Oil thermal conductivity	0.1278		W/(m·K)
Oil-air surface tension	2.45·10 ⁻²		N/m

It is fundamentally important to limit the minimum and maximum allowable temperatures in the solver to ensure convergence of the solution. The limits chosen are the lowest and highest temperatures found in the boundary conditions for the given case, plus an additional margin. In the idle case, the allowed temperature range is $293\text{ K} \leq T \leq 493\text{ K}$, while the MTO case has $373\text{ K} \leq T \leq 543\text{ K}$.

5.6.2 Results at Idle

After an initialisation with constant fluid temperature of 373 K in the whole domain, the average temperature of the annulus dropped in the first iterations of the simulation to the lowest limit of 293 K, likely because of initial convergence issues. The temperature successively increased to stabilise at the final value of 377 K. The achievement of the final stationary temperature took about 1.2 seconds of flow time, as reported in the plot of Figure 5.20. The contours of the wall heat transfer coefficients are compared against the

oil film distributions in Figure 5.21. Analogously to the KIT bearing chamber case (Section 4.5), a clear correspondence is found between the areas with high HTC values and the distribution of thick films on the chamber wall.

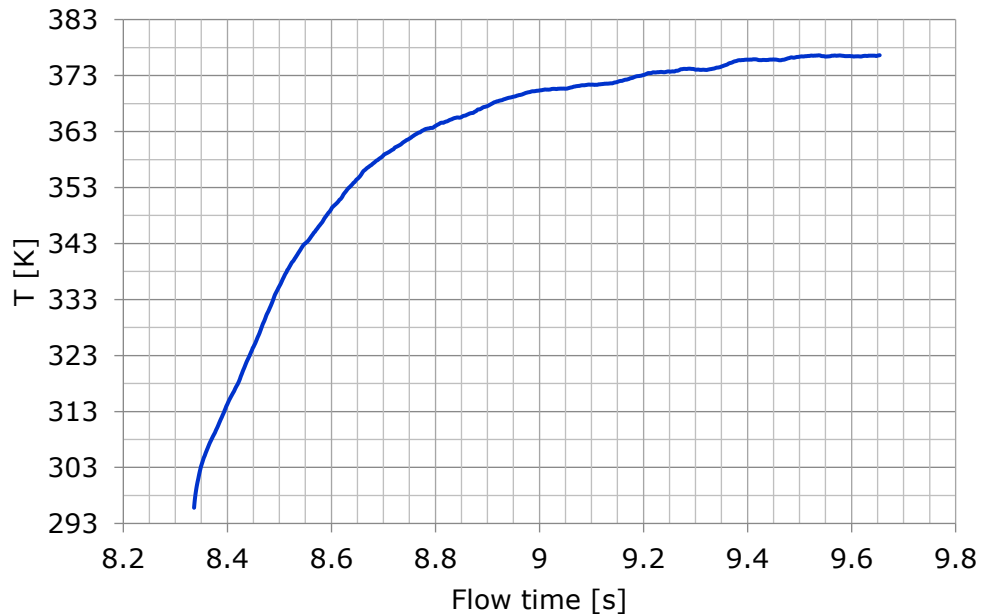


Figure 5.20 – Time-evolution of average annulus temperature at idle

The wall heat transfer coefficients in the regions covered by thick layers of oil are of the order of $5,000 \text{ W}/(\text{m}^2 \text{ K})$, while values of less than $2,000 \text{ W}/(\text{m}^2 \text{ K})$ are found in the areas with low oil coverage, i.e. towards the top of the chamber. It is important to note that in the regions where prism layers are not available, such as on the two axial extremities of the bearing chamber or on the scavenge ramp, the computed HTCs are much lower than the neighbouring regions and present a visible discontinuity. This behaviour is due to the low mesh resolution, which does not enable to accurately resolve the thermal boundary layer (a wall y^+ value of 1 is typically required). Therefore, the results in these low-resolution regions should be regarded carefully, as they are unlikely to be as accurate as the prism layer-covered zones.

Compared to the KIT chamber results of Section 4.5, the wall HTCs in the thick film regions are about $3,000 \text{ W}/(\text{m}^2 \text{ K})$ lower. This result is consistent with the different operating conditions of the two cases, as the BR725 chamber is operating at idle and is characterised therefore by lower oil speeds. This behaviour is found also in the work of Busam et al. [12], who identified an HTC increase of up to $2,500 \text{ W}/(\text{m}^2 \text{ K})$ from the lowest shaft speed (4,000 rpm) to the highest (15,000 rpm) in their experimental bearing chamber.

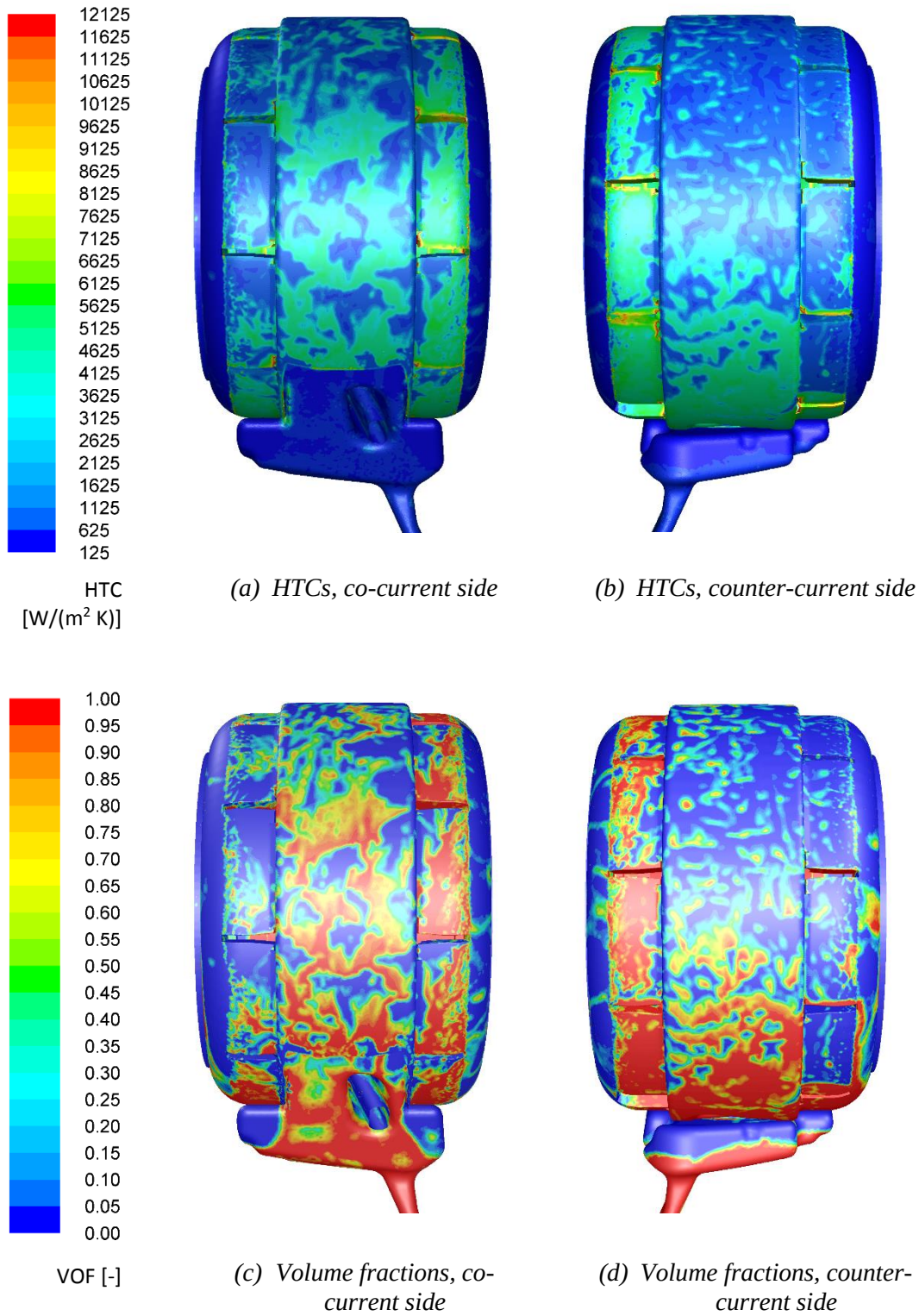


Figure 5.21 – Contours of wall heat transfer coefficients (a, b), and contours of wall volume fractions (c, d) at ground idle , $t=9.64$ s

The time-averaged fluid temperature in the cross-section of the scavenge pipe shown in Figure 5.10 is of 361.7K, against the 373 K from the engine validation tests [135]. The averaging interval is 9.50-9.65 s, covering the stationary temperature range in the plot of Figure 5.20. The lower value found in the simulation is due to the accounting of both air and oil in the temperature monitor, while the validation test data cover the oil temperature only. For this reason, the latter comparison should be considered solely as indicative.

5.6.3 Results at MTO

After the initialisation of the domain with a fluid temperature of 523 K, the average temperature of the air/oil mixture in the annulus decreased to a stable value of 440 K in approximately 0.45 seconds, as shown in the plot of Figure 5.22.

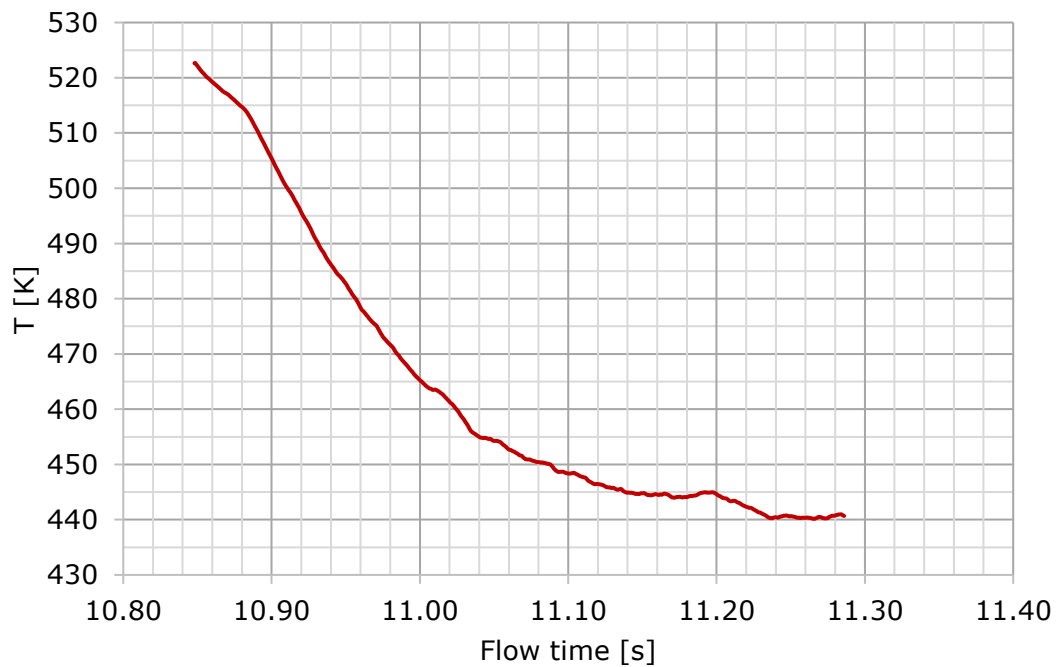


Figure 5.22 – Time-evolution of average annulus temperature at MTO

Compared to the idle flow condition, the annulus temperature stabilises in about one third of the flow time, due to the higher heat exchange that takes place on the bearing chamber walls. This is linked with the higher air flow rates and oil coverage. The plots of the wall heat transfer coefficients, shown in Figures 5.23(a) and (b), confirm the higher overall coverage compared to idle by regions having high HTCs. The result is a higher average HTC value, namely 4,100 W/(m² K) against the 3,500 W/(m² K) computed in the idle case.

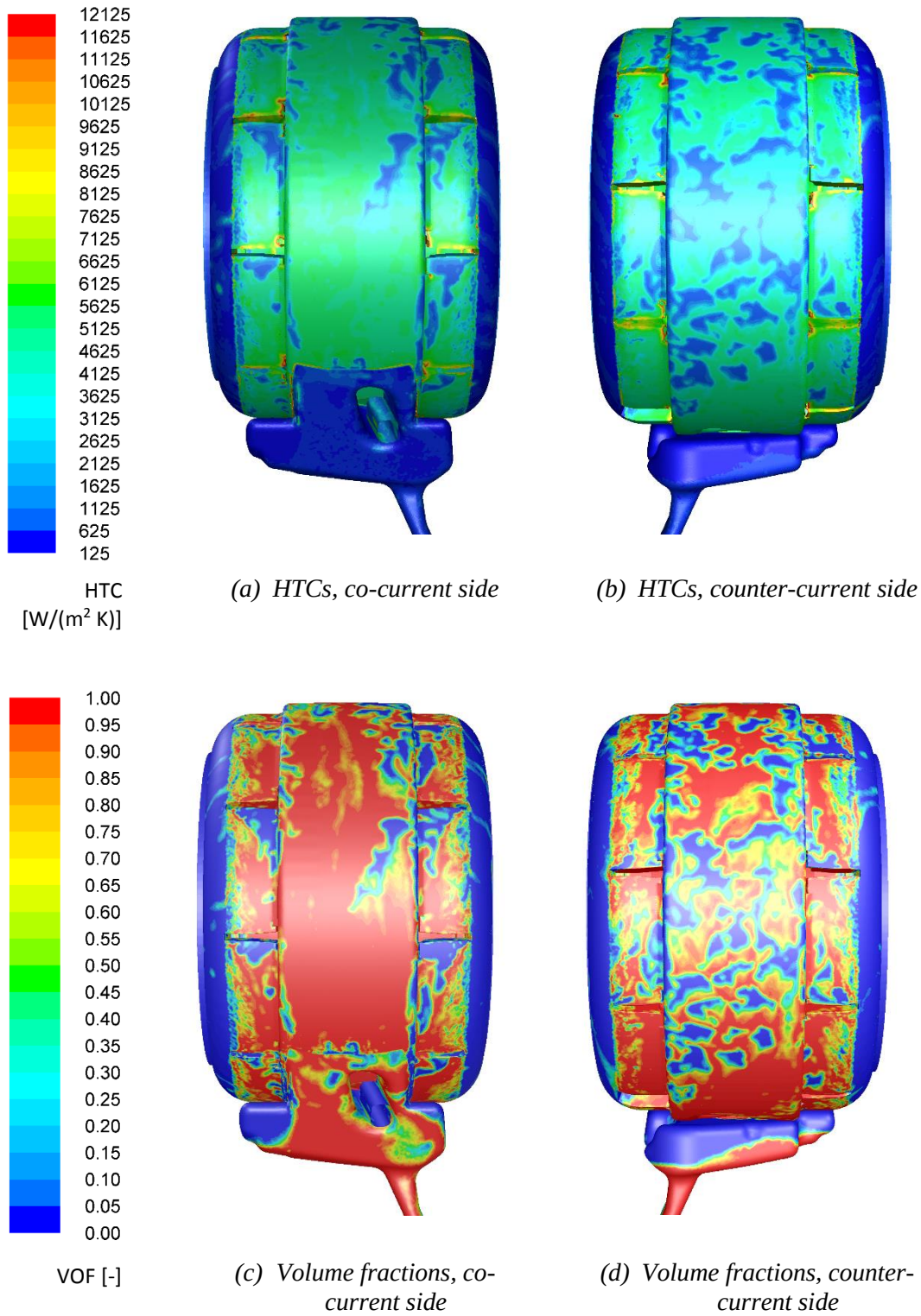


Figure 5.23 – Contours of wall heat transfer coefficients (a, b), and contours of wall volume fractions (c, d) at MTO, $t=11.28$ s

The MTO computed HTC's do not show an increase from idle as significant as expected from the experience on the KIT bearing chamber case (Section 4.5). This is due to the relatively low difference in computed wall film velocity between the two conditions (see Section 5.5.4). A relevant increase is found only in the co-current side of the chamber, where the higher film velocities at MTO give wall HTC's about $1,000 \text{ W}/(\text{m}^2 \text{ K})$ higher. Figures 5.23(c) and (d) report the wall film coverage, showing the clear match between oil film thickness and HTC distribution.

5.7 Summary and Discussion of Results

In this chapter, the Volume of Fluid (VOF) method has been implemented for the first time on the Rolls-Royce BR725 Rear Bearing Chamber (RBC) geometry and operating conditions. The complex geometry required shifting from the conventional structured meshes in favour of polyhedral meshing, enabling to test it for the first time on a geometry of this kind. The simulations included simplified heat transfer models to assess the accuracy and cost of VOF thermal modelling. The main findings include the dominance of the LP shaft in driving the flow, as well as the discovery of a link between the outlet flow dominant frequencies and the flow regime in the chamber annulus.

The lack of previous experience on two-shaft models implied that the relative influence of the shafts in driving the oil flow was initially unknown. The simulations have led to identify the prevalence of the LP shaft over the HP shaft in defining the flow regime. The regime is gravity-driven at low shaft speeds, and has transitional characteristics at MTO. This result is unexpected based on the previous experience on the KIT bearing chamber, where MTO conditions give a shear-driven flow. This finding highlights the importance of the relative portion of shaft in contact with the fluid in defining the flow condition.

The results at idle have shown that oil tends to accumulate at the bottom of the chamber, leading to a thick pool at the base of the jet block. The dominant effect of gravity facilitates oil extraction, giving a high scavenge efficiency value which matches well the design assumption, as well as the reference experimental data from the BR700NG engine. The protruding vent operates effectively in this flow condition, as only a minimal fraction of oil entrains with the air outlet flow.

The oil behaviour at MTO changes significantly compared to idle. A more homogeneous film distribution takes place on the chamber outer wall. The pool previously

found at idle becomes a film about 1 mm thick, although oil still tends to flow counter-current towards the bottom of the chamber. The reasons for this behaviour have been identified in the aerodynamic effect of the jet block, as well as in the low contribution to the overall air momentum given by the HP shaft. Uncertainties related with turbulence damping need to be considered as well, as the experience with the KIT bearing chamber in Chapter 4 has shown that the predictive accuracy for the distribution of highly shear-driven liquid flows is particularly susceptible to the damping method and its scaling (see Section 4.4.4).

The high air shear effects observed at MTO influence also the scavenge efficiency, which is affected by the high rate of sealing air injected in the chamber. Oil surrounding the vent pipe tends to entrain with the fast outgoing air flow despite the protruding geometry, decreasing significantly the scavenge efficiency compared to idle. The CFD model tends to over-predict this effect, giving a value 18.5% lower than the 50% design assumption [135]. However, when comparing the data with the BR700NG tests, this discrepancy is expected to be smaller. These observations confirm the capability of the VOF method to identify the main flow characteristics correctly for both operating conditions. Further tuning of the turbulence damping factor is necessary to improve the match with the experimental data. The computed wall heat transfer coefficients (HTCs) show that there is not a significant change between the idle and MTO operating conditions, mainly because the film surface velocities are of the same order for both cases. The average wall HTCs obtained are of about $3,000 \text{ W}/(\text{m}^2 \text{ K})$. This value is lower than the one found in the KIT bearing chamber simulation, consistently with the lower average oil film velocities.

Computational times have been a critical factor when running the simulation, in particular when including heat transfer. Table 5.9 highlights how the computational costs increase rapidly as the boundary conditions add complexity to the flow. The last column of the table reports the variation over the previous case. The computational costs are nearly doubled when sealing air is injected, as the iterations required for convergence at each time step increase significantly. The implementation of the energy equation gives an additional increase of nearly 50% in CPU cost, corresponding to 3.7 weeks required per second of flow with the computational power available. The largest increase is found at MTO, where the reduction of the time step size, combined with a generally higher number of iterations per time step, gives a 3.5-times increase in computational cost. Fortunately, the latter is

balanced by a shorter flow time required to achieve the flow stationary state, as shown in the plot of Figure 5.8.

Table 5.9 – RBSS simulation computational costs

Case	Time Step [s]	Cores	Sim. Speed [h _{comp} /s _{flow}]	CPU Cost [CPUh/s]	CPU Cost Variation
Zero Rotational Speed	10 ⁻³	192	9.2	1,768	-
Idle, no Sealing Air	10 ⁻⁵	288	218.5	62,928	-
Half-idle with Sealing Air	10 ⁻⁵	288	415.2	119,568	+90.0%
Idle with Heat Transfer	10 ⁻⁵	288	614.0	176,832	+47.9%
MTO	5·10 ⁻⁶	288	2176.7	626,880	+354.5%

The analysis of the computational times indicates that, to date, the Volume of Fluid method is not yet mature for the industrial application on bearing chambers. In order to compute multiple design points, simulation times of the order of few days are required, rather than several weeks. Nonetheless, the method has given an extensive insight for the two operating conditions analysed. Uncertainties related with the modelling of turbulence damping need further work to ensure that the results are reliable also at high shaft speeds. Future research will need to focus on this aspect, as well as on the reduction of computational times, either through coarser grids or more efficient methods.

As heat transfer modelling creates an additional cost for the solver, the definition of a correlation between oil film thickness/velocity and heat transfer coefficients would contribute as well to reduce the computational requirements of the model. Detailed guidelines for future work are proposed in Section 6.3.

5.8 Simulation Guidelines

Model Choice

From the lessons learnt on the KIT bearing chamber simulations of Chapter 4, the recommended choice for the BR725 RBC is the SST k- ω turbulence model, with implicit VOF formulation, Compressive interface reconstruction, and bounded second order implicit time formulation. This set-up gives faster convergence times than the typically

used explicit VOF with Geometric reconstruction, as well as providing a robust solution despite the presence of regions having Courant numbers higher than one, namely in proximity of the air inlets and vent. A critical aspect of the model is the choice of the turbulence damping factor. The value of 50 adopted in the simulation presented in this chapter gave good results at idle, while a higher level of uncertainty affected the MTO case. The recommended choice to date is a damping factor value of 50, although future studies will need to enhance the robustness of this choice.

Mesh

Given the complexity of the geometry to be modelled, an unstructured mesh is the most suitable option for the definition of the computational grid. Early attempts included a tetrahedral and a hex-dominant volume mesh. While the first gave poor convergence properties, the second led to an excessive number of elements required to fill the chamber annulus (of the order of 15+ millions). The optimal solution has been identified in a polyhedral mesh with near-wall prism layers. This ensured good convergence in all the flow conditions tested, while minimising the element count for the chosen sizing options. The maximum element volume for the final mesh is of 4 mm³ in the chamber annulus. A uniform sizing of the elements, without large volume discontinuities from the walls to the core of the fluid region, is recommended.

Prism layers are needed in the regions where oil films are expected to form, i.e. on the chamber outer wall, so that a sufficient resolution through the film depth is retained. Eight layers for a total thickness of about 2 mm have been applied, consistently with an expected film thickness of less than 2 mm. Additional layers should cover the two shafts to ensure a limited range of wall y^+ values on their surface, although the SST k- ω model can provide reliable results also when the prism layers are not present [120]. An additional study to evaluate if the prism layers on the shafts are not needed is recommended, as this would contribute to reduce the mesh size. Nonetheless, the low y^+ requirements of the thermal model should be kept in mind when considering this option.

Time Step

As stated in the Solver Settings section (5.3.5), the time step choice is primarily limited by the mesh size in proximity of the shafts. Courant numbers can exceed the value of one in these regions, although excessive values may lead to poor convergence properties. Values of the order of 10 on the outer regions of the shafts have been found acceptable and not affecting the convergence of the simulation. These correspond to a time step size

of 10^{-5} seconds at ground idle and of $5 \cdot 10^{-6}$ seconds at maximum take-off power. Preliminary testing to identify the optimal time step size is recommended before running a new case.

Initialisation and Simulation Procedure

The domain initialisation has been performed by letting the oil flow from the bearing inlets without any shaft rotation or air injection from the seals, as suggested by Peduto et al. [41]. Consistently with the KIT bearing chamber case of Chapter 4, the tank at the bottom of the scavenge pipe is required to be filled with oil during the simulation.

The step-by-step procedure illustrated in Figure 5.6 was followed to ensure the simulation stability and to assess individually the parameters affecting the flow conditions and oil distributions. It has been found, however, that the solver and mesh are sufficiently robust to change multiple parameters in one time. Therefore, future simulations may be performed by adopting directly the desired boundary conditions configuration, thus saving computational time.

Data Processing

The main recommendations for data processing cover the film thickness measurement on the bearing chamber outer wall. The need to avoid geometrical features such as slots and protrusions requires a careful choice of the location of the film thickness averaging areas. The best approach is to generate a diagram showing the cross-sections in which the measurement areas are going to be located, and to note the angular locations where the latter cannot be placed, as shown in Figures 5.17-5.19. Particular care is required in the slots of the bearing supports, as the post-processing tool may identify oil films that are on the wall opposite to the one investigated. For this reason, an additional check has been implemented in the Matlab post-processing tool. Time averaging is also a critical aspect of post-processing, as the manual exporting of the iso-surfaces of the oil volume fractions can be time-consuming and lead to high memory requirements for the storage of the data. The ideal choice is to use the time-averaging option available in the solver, so that the post-processing can be done from a single iso-surface rather than manually time averaging multiple ones.

6 Conclusions

6.1 Summary

This thesis work demonstrated for the first time the application of the Volume of Fluid method onto the bearing chamber geometry of a production aero-engine. Moving beyond the simplified geometries modelled so far in the literature, the work evaluated the level of accuracy of the current technique on the “real life” case. The aspects of the method useful to support the design, as well as the current limitations, have been identified in this thesis.

The VOF method has shown its main strengths in the robustness of the solution and the adaptability to ideally any complex geometry and flow condition. This has enabled to simulate the bearing chamber flows until the achievement of the stationary state for all the conditions planned. The main limitations have been identified in the long computational times, linked with the transient characteristics of the flow, as well as in the uncertainties from the modelling of the air-oil interaction. Mitigation strategies have been devised for both aspects, including polyhedral meshing, the selection of an appropriate interface reconstruction scheme, and the “tuning” of the turbulence damping function.

The analysis of a two-shaft configuration showed that the shaft rotational speed is not the only parameter determining the flow regime, but it is rather the relative area of the moving walls driving the fluid at a certain rotational speed. To support the investigation of the flow regime during engine testing, this work identified a link between the conditions inside the annulus and the oscillatory characteristics of the scavenge flow. The link suggests that the amplitude of the dominant frequencies in the latter could be used to determine whether the flow is gravity- or shear-dominated, thus indirectly the oil distribution inside the chamber.

The results of the application of VOF have demonstrated the good performance of the method for idle conditions, while at high rotational speeds a higher degree of uncertainty is still linked with its turbulence damping effects. This has led to recommend that this aspect becomes a major focus of research for future applications, as the capacity to predict correctly the flow regime and oil distribution will depend significantly on it. Further details and recommendations for these future research activities are given in Section 6.3.

Despite the high computational cost, the ability to simulate wall-fluid heat transfer with VOF has been demonstrated in this work. From this analysis, the reference experimental observations have been successfully matched by the CFD results.

6.2 Contributions to Knowledge

The unique contributions to knowledge of this work are summarised in the following points:

- The application of VOF with heat transfer onto the geometry of a real bearing chamber has been demonstrated for the first time;
- The main limitation of VOF for bearing chamber applications has been identified in the current turbulence damping approaches;
- A strategy to address the current issues with turbulence damping and to speed-up the computational times has been devised;
- Extensive understanding of the flow characteristics in the BR725 RBC has been achieved through the CFD analysis;
- A link between the scavenge flow characteristics and the bearing chamber flow regime has been identified;
- The deployment of polyhedral meshing on a complex bearing chamber model has been demonstrated.

These contributions are firsts in the field of CFD for bearing chamber flows. From the work done and the lessons learnt, recommendations for future research activities have been defined. They are reported in the following section.

6.3 Recommendations for Future Work

This work identified the need for a robust VOF turbulence damping method to ensure that the accuracy of the results is maintained at high shaft speeds. Therefore, additional research activities are needed to support the correct air/oil interaction modelling for any flow condition. The aim is to avoid the risk of imposing the flow regime rather

than predicting it, as observed in some of the cases analysed in this thesis. Three options are recommended:

- Define a method to compute the turbulence damping factor;
- Address stability issues of the Simplified Interface Treatment [39];
- Investigate/develop alternative turbulence damping approaches.

To address the first point, the gravity parameter λ has been identified in this work as a link between the thin film model predictions and the VOF results. A correlation between the two models could define the level of damping needed to match the local turbulent viscosity at the gas-liquid interface, for any given case. This approach would rely on a preliminary one-dimensional simulation to select the damping factor (B) value required for the 3D VOF simulation. A more sophisticated method could compute the local interface y^+ value within the VOF simulation and, based on a correlation between y^+ and B , apply a local value of B on a cell-by-cell basis.

Despite its simple formulation and the numerical instabilities, the Simplified Interface Treatment has shown promising results on coarse meshes [92], thus it is recommended to investigate it further. The stability issues experienced have been found to be mesh-dependent. An implementation that is more robust than the one currently proposed [39] would enable extensive testing on sufficiently fine meshes.

The investigation of alternative turbulence damping approaches involves additional research in the field of shear-driven flows. A field particularly active on this topic is nuclear industry, where dedicated codes have been developed to model two phase flows (see for example [73, 84]) for reactor cooling problems. The development of alternative turbulence damping approaches is the most costly option in terms of time and resources. Its choice should be weighed carefully against the option of further developing and improving the existing techniques.

The simulations performed in this thesis have shown that long computational times are a serious obstacle to performing parametric studies. This limits the chances of understanding the reasons for a certain flow behaviour. While the numerical cases reproduce the experimental set-ups available, the existing experiments are focused on representing a bearing chamber with all its features rather than for numerical model development. Therefore, too many factors affecting the oil behaviour are present (e.g.

sealing flow, vent flow, oil inlet). A dedicated experimental set-up ruling out the aforementioned disturbances would generate the necessary validation data to investigate purely turbulence damping for shear-driven rimming flows. The suggested set-up is a bearing chamber without inlets or outlets, containing a fixed volume of oil, easily representable in CFD using a limited size mesh.

This research work has explored the applicability of heat transfer modelling on the VOF simulation of bearing chamber flows. The use of the energy equation has led to increase the computational times up to 50% for the real bearing chamber case. Having found that the wall heat transfer coefficients depend mainly on the oil film thickness and velocity, it is recommended to investigate a correlation between heat transfer and liquid flow properties. This is likely to save considerable computational time on future bearing chamber simulations.

The findings of this thesis have been included in the AERIS project, which is part of the EU-funded Clean Sky 2 programme. The techniques adopted have fed also into the MAMOTH project, supporting the development of the Rolls-Royce Power Gearbox. Therefore, it has been ensured that research towards the applicability of VOF to aero-engine bearing chambers will continue and build on the outcome of this work.

7 Literature

- [1] Exxon Mobil Corporation, 2009, "Oil Coking - Part Two: Engine Influences."
- [2] Exxon Mobil Corporation, 2009, "Oil Coking - Part One: The Processes."
- [3] Personal conversation with Mr Paul Rees, Technical Advisor at the Rolls-Royce Transmission UTC, University of Nottingham,
- [4] Florin, P., Von Tunzelmann, N., and Richard, P., 2012, "Asymmetries and dynamic interactive capabilities in technology transfer between ONERA - the French Aerospace LabTM and SMEs," *Journal of Innovation Economics & Management*, 1/2012(9), pp. 103-137.
- [5] Rolls-Royce, 2005, The Jet Engine,
- [6] Kakimpa, B., 2016, "Progress in thin film modelling, 2009-2017," Slideset, The University of Nottingham.
- [7] Jacobs, A.P., Wang, C., Morvan, H.P., Hibberd, S., and Cliffe, K.A., 2012, "ELUBSYS D1.6: Real Engine TBH and KIT Bearing Housing Simulation Results," Report FSG 90660, Rolls-Royce Plc.
- [8] Wang, C., Morvan, H.P., Hibberd, S., and Cliffe, K.A., 2011, "Thin film modelling for aero-engine bearing chambers", Paper GT2011-46259, ASME Turbo Expo 2011,
- [9] Peduto, D., 2015, "Oil Droplet Impact Dynamics in Aero-Engine Bearing Chambers-Correlations derived from Direct Numerical Simulations," PhD Thesis, Karlsruhe Institute of Technology.
- [10] Kurz, W., Dullenkopf, K., and Bauer, H.-J., 2012, "Influences on the oil split between the offtakes of an aero-engine bearing chamber", Paper GT2016-69412, ASME Turbo Expo 2012, Copenhagen, Denmark
- [11] Wittig, S., Glahn, A., and Himmelsbach, J., 1994, "Influence of high rotational speeds on heat transfer and oil film thickness in aero-engine bearing chambers," *Journal of engineering for gas turbines and power*, 116(2), pp. 395-401.
- [12] Busam, S., Glahn, A., and Wittig, S., 2000, "Internal Bearing Chamber Wall Heat Transfer as a Function of Operating Conditions and Chamber Geometry," *ASME Journal of Engineering for Gas Turbines and Power*, 122(2), pp. 314-320.
- [13] Glahn, A., Busam, S., and Wittig, S., 1997, "Local and Mean Heat Transfer Coefficients along the Internal Housing Walls of Aero Engine Bearing Chambers", Paper GT-97-261, ASME Turbo Expo 1997,
- [14] Glahn, A., Kurreck, M., Willmann, M., and Wittig, S., 1996, "Feasibility Study on Oil Droplet Flow Investigations Inside Aero Engine Bearing Chambers—PDPA Techniques in Combination With Numerical Approaches," *Journal of engineering for gas turbines and power*, 118(4), pp. 749-755.
- [15] Glahn, A., and Wittig, S., 1996, "Two-phase air/oil flow in aero engine bearing chambers: characterization of oil film flows," *Journal of engineering for gas turbines and power*, 118(3), pp. 578-583.
- [16] Glahn, A., and Wittig, S., 1999, "Two-Phase Air/Oil Flow in Aero-Engine Bearing Chambers—Assessment of an Analytical Prediction Method for the Internal Wall Heat Transfer," *International Journal of Rotating Machinery*, 5(3), pp. 155-165.
- [17] Glahn, A., and Wittig, S., "Two-Phase Air/Oil Flow in Aero-Engine Bearing Chambers - Assessment of an Analytical Prediction Model for the Internal Wall Heat Transfer," Proc. ISROMAC-6 1996.
- [18] Gorse, P., Willenborg, K., Busam, S., Ebner, J., Dullenkopf, K., and Wittig, S., 2003, "3D-LDA measurements in an aero-engine bearing chamber", Paper GT2003-38376, Proceedings of ASME Turbo Expo 2003, Power for Land, Sea, and Air, Atlanta, Georgia, USA

- [19] Gorse, P., Busam, S., and Dullenkopf, K., 2006, "Influence of operating condition and geometry on the oil film thickness in aeroengine bearing chambers," *Journal of engineering for gas turbines and power*, 128(1), pp. 103-110.
- [20] Kurz, W., and Bauer, H.-J.r., 2014, "An approach for predicting the flow regime in an aero engine bearing chamber", Paper GT2014-26756, ASME Turbo Expo 2014, Düsseldorf, Germany
- [21] Kurz, W., Dullenkopf, K., and Bauer, H., "The impact of geometry variations on the two-phase flows in aero-engine bearing chambers," Proc. 20th ISABE Conference (ISABE 2011), Gothenburg, Sweden, September, pp. 12-16.
- [22] Kurz, W., Dullenkopf, K., and Bauer, H.-J., 2013, "Capacitive Film Thickness Measurements in a Ventless Aero-Engine Bearing Chamber - Influence of Operating Conditions and Offtake Design", Paper GT2013-94973, ASME Turbo Expo 2013, San Antonio, Texas, USA
- [23] von Plehwe, F.C., Krug, M.B., Höfler, C., and Bauer, H.-J., "Experimental and Numerical Investigations on Oil Leakage Across Labyrinth Seals in Aero Engine Bearing Chambers," Proc. ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, American Society of Mechanical Engineers, pp. V001T001A010-V001T001A010.
- [24] Chandra, B., 2012, "Parametric Study of Shallow AE3007 Sump," Report UTC JF24/BC/04, The University of Nottingham
- [25] Chandra, B., Collicott, S.H., and Munson, J.H., 2013, "Scavenge Flow in a Bearing Chamber With Tangential Sump Off-Take," *Journal of Engineering for Gas Turbines and Power*, 135(3), p. 032503.
- [26] Chandra, B., and Simmons, K., 2016, "Innovative shallow sump customizations for aero-engine bearing chambers", Paper GT2016-56107, ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, Seoul, South Korea
- [27] Chandra, B., Simmons, K., Pickering, S., Colliocot, S.H., and Wiedemann, N., 2012, "Study of Gas/Liquid behaviour within an aeroengine bearing chamber", Paper GT2012-68753, ASME Turbo Expo 2012, Copenhagen, Denmark
- [28] Chandra, B., Simmons, K., Pickering, S., and Tittel, M., 2011, "Liquid and gas flow behaviour in highly rotating environment", Paper GT2011-46430, Proceedings of ASME Turbo Expo 2011, Vancouver, BC, Canada
- [29] Chandra, B., Simmons, K., Pickering, S., and Tittel, M., 2010, "Factors affecting oil removal from an aeroengine bearing chamber", Paper GT2010-22631, ASME Turbo Expo 2010,
- [30] Simmons, K., and Chandra, B., 2014, "Experimental investigation into the performance of shallow aeroengine off-takes", Paper GT2014-26132, ASME Turbo Expo 2014, Düsseldorf, Germany
- [31] Jingyu, Z., and Zhenxia, L., 2015, "Numerical and Experimental Study for Unsteady Oil Film Thickness of the Rotating Cylinder Chamber Wall," *Journal of Engineering for Gas Turbines and Power*, 137(12), p. 122501.
- [32] Hashmi, A.A., Dullenkopf, K., Bauer, H.-J.r., and Klingsporn, M., 2011, "Experimental Investigation of Lubrication Oil Film Dynamics in a Typical Aero-Engine Bearing Chamber Environment", Paper GT2011-45545, ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition, Vancouver, Canada
- [33] Gorse, P., Dullenkopf, K., Bauer, H.-J., and Wittig, S., "An experimental study on droplet generation in bearing chambers caused by roller bearings," Proc. ASME Turbo Expo 2008: Power for Land, Sea, and Air, American Society of Mechanical Engineers, pp. 1681-1692.
- [34] Krug, M.B., Höfler, C., Bauer, H.-J., and Kurz, W., 2015, "Evaluation of the Volume-of-Fluid Method for the Numerical Modeling of an Aero Engine Bearing

Chamber", Paper ISABE-2015-21248, The 22nd International Symposium on Air Breathing Engines, Phoenix, AZ, USA

[35] Aidarinis, J., Missirlis, D., Yakinthos, K., and Goulas, A., 2011, "CFD modeling and LDA measurements for the air-flow in an aero engine front bearing chamber," *Journal of Engineering for Gas Turbines and Power*, 133(8), p. 082504.

[36] Chew, J.W., 1996, "Analysis of the Oil Film on the Inside of an Aero-Engine Bearing Chamber Housing", Paper 96-GT-300, ASME Turbo Expo 1996,

[37] Kay, E., 2011, "Non-isothermal Bearing Chamber Modelling: an Enhanced Lubrication Theory Model," Report UTC FF69/EDK/03, The University of Nottingham

[38] Bristot, A., Personal conversation with Dr Wolfram Kurz,

[39] Hashmi, A.A., 2012, "Oil Film Dynamics in Aero Engine Bearing Chambers - Fundamental Investigations and Numerical Modelling," PhD Thesis, Karlsruhe Institute of Technology.

[40] Hashmi, A.A., Dullenkopf, K., Koch, R., and Bauer, H.-J.r., 2010, "CFD methods for shear driven liquid wall films", Paper GT2010-23532, ASME Turbo Expo 2010, Glasgow, UK

[41] Peduto, D., Hashmi, A.A., Dullenkopf, K., Bauer, H.J.r., and Morvan, H., 2011, "Modelling of an Aero-Engine Bearing Chamber Using Enhanced CFD Technique", Paper GT2011-45635, ASME 2011 Turbo Expo, Vancouver, Canada

[42] Tkaczyk, P., 2011, "CFD simulation of annular flows through bends," PhD Thesis, The University of Nottingham.

[43] Tkaczyk, P., and Morvan, H.P., 2012, "Methodology for Modelling Two-Phase Flow in Bearing Chambers using CFD (Volume of Fluid) — SILOET," Report UTC JF82/PT/06, The University of Nottingham

[44] Crouchez-Pillot, A., and Morvan, H.P., 2014, "CFD Simulation of an Aero engine bearing chamber using an Enhanced Volume of Fluid (VOF) Method", Paper GT2014-26405, ASME Turbo Expo 2014, Düsseldorf, Germany

[45] Adeniyi, A., 2015, "A Coupled Lagrangian-Eulerian Framework to model droplet to film interaction with Heat Transfer," PhD Thesis, University of Nottingham.

[46] Adeniyi, A., 2014, "A transient CFD simulation of the flow in a test rig of an aeroengine bearing chamber", Paper GT2014-26399, ASME Turbo Expo 2014, Düsseldorf, Germany

[47] Adeniyi, A., Morvan, H., and Simmons, K., 2013, "Droplet-Film Impact Modelling Using a Coupled DPM-VoF Approach", Paper ISABE-2013-1419, 21st Int. Soc. for Air Breathing Engines (ISABE), Busan, Korea

[48] Bristot, A., Personal communication with Dr Bruce Kakimpa, November 2016

[49] Farrall, M., 2000, "Numerical modelling of two-phase flow in a simplified bearing chamber," PhD Thesis, University of Nottingham.

[50] Kay, E., Hibberd, S., and Power, H., 2014, "A depth-averaged model for non-isothermal thin-film rimming flow," *International Journal of Heat and Mass Transfer*, 70, pp. 1003-1015.

[51] Kakimpa, B., Morvan, H., and Hibberd, S., 2016, "The depth-averaged numerical simulation of laminar thin-film flows with capillary waves," *Journal of Engineering for Gas Turbines and Power*, 138(11), p. 112501.

[52] Baxter, S.J., 2010, "Numerical Simulation Of Three-Dimensional Free Surface Film Flow Over Or Around Obstacles On An Inclined Plane," PhD Thesis, University of Nottingham.

[53] Williams, J., 2008, "Thin film rimming flow subject to droplet impact at the surface," PhD Thesis, University of Nottingham.

- [54] Wang, C., and Morvan, H.P., 2011, "Thin Film Formulation and Implementation in a CFD Code for Oil Flow Simulations of Bearing Chambers " Report ELUBSYS WP2 Deliverable, The University of Nottingham
- [55] Formenti, A., Emamzadeh, M., and Morvan, H.P., 2012, "Advanced Modelling of Heat Transfer and Oil/Air Flows in Bearing Chambers: Report on Trent-style bearing chamber model," Report UTC JF59/AF/04, The University of Nottingham
- [56] ERCOFTAC, 2014, "SPHERIC Home Page," https://wiki.manchester.ac.uk/spheric/index.php/SPHERIC_Home_Page.
- [57] Gingold, R.A., and Monaghan, J.J., 1977, "Smoothed particle hydrodynamics: theory and application to non-spherical stars," *Monthly notices of the royal astronomical society*, 181(3), pp. 375-389.
- [58] Morvan, H.P., 2010, "Review of CFD Modelling Approaches for Bearing Chambers," Report ELUBSYS D2.1, The University of Nottingham
- [59] Kruisbrink, A., Pearce, F., Yue, T., Cliffe, A., and Morvan, H., 2011, "SPH concepts for continuous wall and pressure boundaries," Proc. The 6th International SPHERIC workshop, Hamburg, Germany, pp. 298-304.
- [60] Kruisbrink, A.C.H., Morvan, H.P., and Pearce, F.R., 2014, "Progress in Smoothed Particle Hydrodynamics to Simulate Bearing Chambers", Paper GT2014-26403, ASME Turbo Expo, Düsseldorf, Germany
- [61] Korsukova, E., Kruisbrink, A., Morvan, H., Paleo Cageao, P., and Simmons, K., 2016, "Oil Scoop Simulation and Analysis Using CFD and SPH", Paper GT2016-57554, ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, Seoul, South Korea
- [62] Korzilius, S., Kruisbrink, A., Schilders, W., Anthonissen, M., and Yue, T., 2014, "Momentum conserving methods that reduce particle clustering in SPH," Report CASA; Vol. 1415, Technische Universiteit Eindhoven, Eindhoven
- [63] Wieth, L., Lieber, C., Kurz, W., Braun, S., Koch, R., and Bauer, H.J.r., 2015, "Numerical modeling of an aero-engine bearing chamber using the meshless Smoothed Particle Hydrodynamics method", Paper GT2015-42316, Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition, Montréal, Canada
- [64] Hirt, C.W., and Nichols, B.D., 1981, "Volume of fluid (VOF) method for the dynamics of free boundaries," *Journal of computational physics*, 39(1), pp. 201-225.
- [65] Pilliod Jr, J.E., and Puckett, E.G., 2004, "Second-order accurate volume-of-fluid algorithms for tracking material interfaces," *Journal of Computational Physics*, 199(2), pp. 465-502.
- [66] Unverdi, S.O., and Tryggvason, G., 1992, "A front-tracking method for viscous, incompressible, multi-fluid flows," *Journal of computational physics*, 100(1), pp. 25-37.
- [67] Popinet, S., and Zaleski, S., 1999, "A front-tracking algorithm for accurate representation of surface tension," *International Journal for Numerical Methods in Fluids*, 30(6), pp. 775-793.
- [68] Bourlioux, A., "A coupled level-set volume-of-fluid algorithm for tracking material interfaces," Proc. Proceedings of the 6th International Symposium on Computational Fluid Dynamics, Lake Tahoe, CA, 15.
- [69] Sussman, M., and Puckett, E.G., 2000, "A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows," *Journal of Computational Physics*, 162(2), pp. 301-337.
- [70] ANSYS, 2016, Fluent v17 User Manual, Boulder, CO, USA, www.ansys.com.
- [71] Seo, J., 2014, "Multiphase Flow Modeling with Free Surfaces Flow," PDF file, <http://simplorer.com/Resource+Library/Presentations/Multiphase+Flow+Modeling+with+Free+Surfaces+Flow+-+Presentation>.

- [72] Long, R.R., 1970, "Film notes for stratified flow," Report No. 21618, National Committee for Fluid Mechanics Films, web.mit.edu/hml/ncfmf/18SF.pdf.
- [73] Höhne, T., and Mehlhoop, J.-P., 2014, "Validation of closure models for interfacial drag and turbulence in numerical simulations of horizontal stratified gas–liquid flows," *International Journal of Multiphase Flow*, 62, pp. 1-16.
- [74] Akai, M., Inoue, A., and Aoki, S., 1981, "The prediction of stratified two-phase flow with a two-equation model of turbulence," *International Journal of Multiphase Flow*, 7(1), pp. 21-39.
- [75] Celik, I., and Rodi, W., 1984, "Simulation of free-surface effects in turbulent channel flows," *Physicochemical Hydrodynamics*, 5(3-4), pp. 217-227.
- [76] Benkirane, R., Line, A., and Masbernat, L., "Modelling of wavy stratified flow in a rectangular channel," Proc. ICHMT International Seminar on Phase-Interface Phenomena in Multiphase Flow, Dubrovnik, 14, p. 18.
- [77] Murata, A., Hihara, E., and Saito, T., 1991, "Turbulence below an air–water interface in a horizontal channel," *ASME Proceedings of Turbulence Modification in Multiphase Flows, FED*, 110, pp. 67-74.
- [78] Lorencez, C., Chang, T., and Kawaji, M., 1991, "Investigation of turbulent momentum transfer at a gas-liquid interface in a horizontal countercurrent stratified flow," Proc. Turbulence Modification in Multiphase Flows, 110, pp. 97-102.
- [79] Lorencez, C., Nasr-Esfahany, M., Kawaji, M., and Ojha, M., 1997, "Liquid turbulence structure at a sheared and wavy gas-liquid interface," *International journal of multiphase flow*, 23(2), pp. 205-226.
- [80] Egorov, Y., 2004, "Validation of CFD codes with PTS-relevant test cases," Report EVOL-ECORA-D07, ECORA, [https://domino.grs.de/ecora/ecora.nsf/.../\\$FILE/D07_WP4.pdf](https://domino.grs.de/ecora/ecora.nsf/.../$FILE/D07_WP4.pdf).
- [81] Vallée, C., Höhne, T., Prasser, H.-M., and Sühnel, T., 2008, "Experimental investigation and CFD simulation of horizontal stratified two-phase flow phenomena," *Nuclear Engineering and Design*, 238(3), pp. 637-646.
- [82] Deendarlianto, Höhne, T., Lucas, D., Vallée, C., and Zabala, G.A.M., 2011, "CFD studies on the phenomena around counter-current flow limitations of gas/liquid two-phase flow in a model of a PWR hot leg," *Nuclear Engineering and Design*, 241(12), pp. 5138-5148.
- [83] Deendarlianto, Höhne, T., Apanasevich, P., Lucas, D., Vallée, C., and Beyer, M., 2012, "Application of a new drag coefficient model at CFD-simulations on free surface flows relevant for the nuclear reactor safety analysis," *Annals of Nuclear Energy*, 39(1), pp. 70-82.
- [84] Höhne, T., Deendarlianto, and Apanasevich, P., 2013, "New Aspects in the Numerical Modeling of Horizontal Two-Phase Flows in Industrial Applications", Paper FEDSM2013-16022, ASME 2013 Fluids Engineering Division Summer Meeting,
- [85] Bartosiewicz, Y., Seynhaeve, J.-M., Vallée, C., Höhne, T., and Laviéville, J.-M., 2010, "Modeling free surface flows relevant to a PTS scenario: Comparison between experimental data and three RANS based CFD-codes. Comments on the CFD-experiment integration and best practice guideline," *Nuclear Engineering and Design*(240), pp. 2375-2381.
- [86] Bartosiewicz, Y., Laviéville, J.-M., and Seynhaeve, J.-M., 2008, "A first assessment of the NEPTUNE_CFD code: Instabilities in a stratified flow comparison between the VOF method and a two-field approach," *International Journal of Heat and Fluid Flow*(29), pp. 460-478.
- [87] Fabre, J., Masbernat, L., and Suzanne, C., 1987, "Stratified Flow, Part I: Local Structure," *Multiphase Science and Technology*, 3.

- [88] Fernandino, M., and Ytrehus, T., 2008, "Effect of Interfacial Waves on Turbulence Structure in Stratified Duct Flows," *Journal of Fluids Engineering*, 130(061201), pp. 1-8.
- [89] Bristot, A., Morvan, H.P., and Simmons, K.A., 2016, "Evaluation of a Volume of Fluid CFD Methodology for the Oil Film Thickness Estimation in an Aero-engine Bearing Chamber", Paper GT2016-56237, Proceedings of ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, Seoul, South Korea
- [90] Bristot, A., Morvan, H.P., Simmons, K.A., and Klingsporn, M., 2017, "Effect of Turbulence Damping in VOF Simulation of an Aero-Engine Bearing Chamber", Paper GT2017-63436, Proceedings of ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, Charlotte, NC, USA
- [91] Young, C., and Chew, J.W., 2005, "Evaluation of the volume of fluid modelling approach for simulation of oil/air system flows", Paper GT2005-68861, ASME Turbo Expo 2005,
- [92] Krug, M., 2012, "Evaluation of the Volume of Fluid Method for the Numerical Modelling of an Aero Engine Bearing Chamber," Dipl.-Ing Thesis, Karlsruhe Institute of Technology, Karlsruhe, Germany.
- [93] Crouchez-Pillot, A., 2013, "CFD Simulation of an Aero engine bearing chamber using an Enhanced Volume of Fluid (VOF) Method," Rapport final de Travail de Fin d'Etude, The University of Nottingham - Ecole Centrale de Lyon.
- [94] Yaguo, L., Jingyu, Z., Zhenxia, L., and Guozhe, R., 2016, "Numerical study on the effect of oil removal from aero-engine bearing chamber and improvement", Paper 2016 STLE Annual Meeting & Exhibition, Las Vegas, Nevada, USA
- [95] Farrall, M., Hibberd, S., and Simmons, K., 2008, "The effect of initial injection conditions on the oil droplet motion in a simplified bearing chamber," *Journal of Engineering for Gas Turbines and Power*, 130(1), p. 012501.
- [96] Adeniyi, A.A., Morvan, H., and Simmons, K., 2016, "Oil-Air Flow Between the Cage and Inner Race of an Aeroengine Bearing", Paper GT2016-56927, ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition,
- [97] Adeniyi, A.A., Morvan, H., and Simmons, K., 2017, "A Computational Fluid Dynamics Simulation of Oil–Air Flow Between the Cage and Inner Race of an Aero-engine Bearing," *Journal of Engineering for Gas Turbines and Power*, 139(1), p. 012506.
- [98] Versteeg, H.K., and Malalasekera, W., 2007, An introduction to computational fluid dynamics: the finite volume method, Pearson Education,
- [99] Rider, W.J., and Kothe, D.B., 1998, "Reconstructing volume tracking," *Journal of computational physics*, 141(2), pp. 112-152.
- [100] Muzaferija, S., Peric, M., Sames, P., and Schellin, T., "A two-fluid Navier-Stokes solver to simulate water entry," Proc. 22nd symposium on naval hydrodynamics, Washington, DC, pp. 277-289.
- [101] Osher, S., and Sethian, J.A., 1988, "Fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulations," *Journal of computational physics*, 79(1), pp. 12-49.
- [102] Olsson, E., Kreiss, G., and Zahedi, S., 2007, "A conservative level set method for two phase flow II," *Journal of Computational Physics*, 225(1), pp. 785-807.
- [103] Ubbink, O., and Issa, R.I., 1999, "Method for capturing sharp fluid interfaces on arbitrary meshes," *J. Comput. Phys.*, 153, pp. 26-50.
- [104] Waclawczyk, T., and Koronowicz, T., 2008, "Comparison of CICSAM and HRIC high-resolution schemes for interface capturing," *Journal of theoretical and applied mechanics*, 46(2), pp. 325-345.
- [105] Wilcox, D.C., 1998, Turbulence modeling for CFD, DCW industries La Canada, CA,

- [106] Boussinesq, J., 1877, "Théorie de l'écoulement tourbillant," *Mem. Présentés par Divers Savants Acad. Sci. Inst. Fr.*, 23, pp. 46-50.
- [107] Spalart, P.R., and Allmaras, S.R., 1994, "A one-equation turbulence model for aerodynamic flows," *La Recherche Aéronautique*(1), pp. 5-21.
- [108] Spalart, P., and Shur, M., 1997, "On the sensitization of turbulence models to rotation and curvature," *Aerospace Science and Technology*, 1(5), pp. 297-302.
- [109] Smagorinsky, J., 1963, "General Circulation Experiments with the Primitive Equations. I. The Basic Experiment," *Mon. Weather Rev.*, 91(3), pp. 99-164.
- [110] Schumann, U., 1975, "Subgrid Scale Model for Finite Difference Simulations of Turbulent Flows in Plane Channels and Annuli," *J. Comput. Phys.*, 18, pp. 376-404.
- [111] Bardina, J., Ferziger, J.H., and Reynolds, W.C., 1980, "Improved Subgrid-scale Models for Large-eddy Simulation", Paper AIAA 80-1357,
- [112] Germano, M., 1986, "A Proposal for a Redefinition of the Turbulent Stresses in the Filtered Navier-Stokes Equations," *Phys. Fluids*, 29, pp. 2223-2324.
- [113] Argyropoulos, C., and Markatos, N., 2015, "Recent advances on the numerical modelling of turbulent flows," *Applied Mathematical Modelling*, 39(2), pp. 693-732.
- [114] Spalart, P.R., Jou, W.-H., Strelets, M., and Allmaras, S.R., 1997, "Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach," *Advances in DNS/LES*, Creyden Press, Columbus, OH, pp. 137-147.
- [115] Launder, B.E., and Spalding, D., 1974, "The numerical computation of turbulent flows," *Computer methods in applied mechanics and engineering*, 3(2), pp. 269-289.
- [116] Launder, B.E., Morse, A., Rodi, W., and Spalding, D., "Prediction of free shear flows: a comparison of the performance of six turbulence models," *Proc. Free turbulent shear flows*, I, pp. 361-426, 697-698.
- [117] Forster, D., Nelson, D.R., and Stephen, M.J., 1977, "Large-distance and long-time properties of a randomly stirred fluid," *Physical Review A*, 16(2), pp. 732-749.
- [118] Yakhot, V., and Orszag, S.A., 1986, "Renormalization Group Analysis of Turbulence. I. Basic Theory," *Journal of Scientific Computing*, 1(1), pp. 3-51.
- [119] Menter, F.R., 1993, "Zonal two equation k-turbulence models for aerodynamic flows," *ALAA paper*, 2906, p. 1993.
- [120] Menter, F., Kuntz, M., and Langtry, R., 2003, "Ten years of industrial experience with the SST turbulence model," *Turbulence, heat and mass transfer*, 4(1), pp. 625-632.
- [121] Morini, M., Pinelli, M., Spina, P.R., and Venturini, M., 2010, "Computational fluid dynamics simulation of fouling on axial compressor stages," *Journal of Engineering for Gas Turbines and Power*, 132(7), p. 072401.
- [122] Jakoby, R., Zierer, T., Lindblad, K., Larsson, J., Bohn, D.E., Funcke, J., and Decker, A., 2004, "Numerical simulation of the unsteady flow field in an axial gas turbine rim seal configuration", Paper GT2004-53829, ASME Turbo Expo 2004: Power for Land, Sea, and Air,
- [123] Chilton, T.H., and Colburn, A.P., 1934, "Mass transfer (absorption) coefficients - Prediction from data on heat transfer and fluid friction," *Industrial & engineering chemistry*, 26(11), pp. 1183-1187.
- [124] Greitzer, E.M., Spakovszky, Z.S., and Waitz, I.A., 2007, "Thermodynamics and Propulsion notes," <http://web.mit.edu/16.unified/www/FALL/thermodynamics/notes/notes.html>.
- [125] Robinson, A., Morvan, H., and Eastwick, C., 2008, "Computational Investigations Into Aero-Engine Bearing Chamber Off-Take Flows", Paper GT2008-50634, ASME Turbo Expo 2008: Power for Land, Sea, and Air, Berlin, Germany
- [126] Brackbill, J., Kothe, D.B., and Zemach, C., 1992, "A continuum method for modeling surface tension," *Journal of computational physics*, 100(2), pp. 335-354.

- [127] ANSYS, 2015, "Multiphase Modeling using ANSYS Fluent - Lecture 3: Volume of Fluid Model (VOF)," Report, ANSYS, Inc., Boulder, CO, USA
- [128] Kartuzova, O., and Kassemi, M., 2011, "Modeling interfacial turbulent heat transfer during ventless pressurization of a large scale cryogenic storage tank in microgravity", Paper AIAA 2011-6037, 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, San Diego, California
- [129] Tanner, L., and Blows, L., 1976, "A study of the motion of oil films on surfaces in air flow, with application to the measurement of skin friction," *Journal of Physics E: Scientific Instruments*, 9(3), p. 194.
- [130] Ahumada Sanhueza, C., Bristot, A., Kumar, S., Schneider, N., Garvey, S., and Morvan, H., 2016, "An Integrated System's Approach Towards Aero Engine Subsystems Design", Paper SAE 2016-01-2020,
- [131] Piazero de Carvalho, T., 2016, "Two-phase flow in open-cell metal foams with application to aero-engine separators," PhD Thesis, The University of Nottingham, Nottingham, UK.
- [132] Peric, M., and Ferguson, S., 2004, "The advantage of polyhedral meshes," http://www.plmmarketplace.com/upload/Temp/The_Advantage_of_polyhedral_1.pdf.
- [133] International TechneGroup Incorporated, 2017, "CadFix overview," Milford, Ohio, USA, <http://www.iti-global.com/cadfix>.
- [134] ANSYS, 2016, "ANSYS ICEM v17 User Manual," Report, www.ansys.com.
- [135] Personal conversation with Dr M. Klingsporn at Rolls-Royce Deutschland, August 2016,
- [136] Bristot, A., Personal communication with Dr Michael Klingsporn, Oil, Fuel and Heat Management Systems at Rolls-Royce Deutschland, 29/11/2016
- [137] Bristot, A., Personal communication with Dr Thierry Sibilli, Assistant Professor at Rolls-Royce University Technology Centre in Thermal Management, Pusan National University, Busan, Korea, 02/11/2016